

1N-20-CR  
188756

**Final Report**

**Advanced Space Propulsion System Flowfield Modeling**

**Contract NAS8-40845**

**7 February 1998**

**Prepared for**

**National Aeronautics and Space Administration  
George C. Marshall Space Flight Center  
Marshall Space Flight Center, AL 35812**

**By**

**Sheldon D. Smith**

**Huntsville Sciences Corporation  
7525 S. Memorial Parkway Suite D  
Huntsville, AL 35802**

## **FOREWORD**

The following report prepared for NASA/MSFC entitled " Advanced Space Propulsion System Flowfield Modeling" constitutes the final documentation for contract NAS8-40845.

All inquiries regarding this Final Report should be directed to:

**Sheldon D. Smith**  
**Huntsville Sciences Corporation**  
**7525 South Memorial Parkway, Suite D**  
**Huntsville, AL 35802**

TABLE OF CONTENTS

FOREWORD..... II

TABLE OF CONTENTS..... III

1.0 INTRODUCTION AND SUMMARY ..... 1

2.0 DISCUSSION ..... 2

2.1 DESCRIPTION OF THE IMPLIMENTATION OF THE VNAP2 MODEL INTO THE PEP SYSTEM ..... 2

2.2 USING THE VNAP2 MODULE IN PEP ..... 4

2.3 DESCRIPTION OF THE VNAP2 FLOWFIELD MODULE INPUT VARIABLES..... 9

2.4 PEP/VNAP2 FLOWFIELD MODEL VERIFICATION/DEMONSTRATION ..... 26

2.4.1 VERIFICATION CASE 1 ..... 26

2.4.2 DEMONSTRATION CASE 1..... 27

3.0 CONCLUSIONS AND RECOMMENDATIONS..... 40

4.0 REFERENCES..... 40

## 1.0 INTRODUCTION AND SUMMARY

Solar thermal upper stage propulsion systems currently under development utilize small low chamber pressure/high area ratio nozzles. Consequently, the resulting flow in the nozzle is highly viscous, with the boundary layer flow comprising a significant fraction of the total nozzle flow area. Accurate characterization of the nozzle/plume flowfields for these type systems are necessary to support testing, performance characterization and plume induced environments such as plume impingement heating and pressure loads. The conventional uncoupled flow methods which treat the nozzle boundary layer and inviscid flowfield separately by combining the two calculations via the influence of the boundary layer thickness on the inviscid flowfield are not accurate enough to adequately treat highly viscous nozzles. Models such as VNAP2 (1) and VIPER (2) utilize Navier Stokes (NS) and Parabolized Navier Stokes (PNS) methodologies to solve the complete nozzle flowfield in a coupled formulation such that the advanced space propulsion system nozzle flowfields can be accurately treated. However, these codes cannot treat the vacuum plume flowfields. This study built upon recently developed artificial intelligence methods and user interface methodologies to couple the appropriate VNAP2 nozzle flowfield solutions with the RAMP2 vacuum plume code. The RAMP2 (3) plume model that currently is a part of the Plume Environment Prediction Code (PEP, 4) can accurately treat the vacuum exhaust plume characterization but treats the nozzle/boundary interaction via a uncoupled approach. This study added the capability to calculate highly viscous nozzle and plume flowfields into the PEP model via the following specific tasks:

- 1) Develop Artificial Intelligence (AI) analogs for VNAP2
- 2) Develop user interfaces for input and operation of the VNAP2 module within the PEP environment
- 3) Integrate the VNAP2 into the PEP system
- 4) Validation of the VNAP2/PEP flowfield model

This report summarizes the results of this study including:

- 1) Description of the implementation of the VNAP2 module into the PEP system
- 2) Using the VNAP2 module in PEP
- 3) Description of the VNAP2 module input variables
- 4) PEP/VNAP2 flowfield model validation/demonstration

## 2.0 DISCUSSION

### 2.1 Description of the Implementation of the VNAP2 Module into the PEP System

The VNAP2 computer code was developed by Los Alamos National Laboratory for calculating turbulent (as well as laminar and inviscid), steady and unsteady flow. VNAP2 solves the two-dimensional (2D), axisymmetric, time-dependent, compressible Navier-Stokes equations by second-order-accurate finite-difference method. VNAP2 allows arbitrary grid spacing, has two options to speed up calculations for high Reynolds number flows, contains three different turbulence models, and can solve either single- or dual-flowing stream geometry's. This last option allows the VNAP2 code to compute internal/external flows, such as inlets and jet-powered afterbodies as well as airfoils. It is apparent from this general description of the VNAP2 code that it has a wide range of capabilities and as a result input variables. The object of this study was to use the VNAP2 code to calculate highly viscous nozzle flows and subsequently perform a vacuum plume solution using the RAMP2 module of the PEP model. The effort required to generate the logic for input and modification of all the type problems this code can handle is far beyond the scope and resources of this study. Therefore, the input generation/modification module that was developed under this study was limited to automatically generating the VNAP2 input files for performing a combustion/nozzle calculation only. This restriction on the use of the PEP code to generate input data and run the VNAP2 code does not prevent a user from generating an input file and running VNAP2 for any other type of problems. The user merely needs to set up a new named problem, manually generate the VNAP2 module input file \*inp.vnp (where \* is the problem name) and execute the VNAP2 module. The output for the case will be found in the file \*out.vnp.

The basic philosophies that were adopted in implementing a VNAP2 code into the PEP model are:

- 1) The input data file for generating the VNAP2 viscous nozzle solution is generated automatically. A limited number of VNAP2 input variables can be modified using a GUI module.
- 2) The RAMP2 module is utilized to calculate the exhaust plume.
- 3) Nozzle geometry and operating conditions are input using the RAMP2 input module
- 4) Thermodynamic and transport property data required by the VNAP2 code are supplied by the CEC module. The user does not necessarily have to utilize the CEC module but the correct/appropriate transport data must be available to be input through VNAP2 input generation GUI module.

- 5) The RAMP2 nozzle flow solution module must be run prior to generating the VNAP2 input data file and nozzle flow solution. The RAMP2 flow solution provides input data to the automatic input data generation routines via RAMP2 generated data files(\*str.ram and \*lip.ram).
- 6) VNAP2 is an ideal gas model that requires the RAMP2 plume solution utilize the ideal gas variable total enthalpy using average specific and molecular weights.
- 7) Only combustion/chamber nozzle flow solutions can be automatically input and calculated with the VNAP2 module. In the event that the user does not have or input the combustion chamber geometry and as a result inputs the nozzle geometry from the throat, the input generation routine will provide the combustion chamber geometry assuming: the combustion chamber diameter is twice the throat diameter, the inlet angle is 45 degrees, the throat upstream radius of curvature is twice the throat radius, and the combustion chamber to inlet transition radius is twice the throat radius. The user does not have an option of changing these defaults in the event that no combustion chamber geometry has been input.
- 8) In the event that the user enters the VNAP2 input generation module and the code detects the preexisting input file the code will allow the user to use the preexisting file, modify the file using the GUI module or automatically regenerate the input data file.

Based on the above assumptions and philosophies the following modifications/additions were made to the PEP and VNAP2 codes:

- A VNAP2 control subroutine was developed to control input and execution of the VNAP2 module.
- A VNAP2 automatic input file generation routine was added to the PEP model.
- A graphical user interface was developed to allow updates to the VNAP2 input data file.
- VNAP2 was modified to:
  - Data file are named and opened according to the PEP file system convention.
  - VNAP2 generates an output file containing the exit plane data required by RAMP2 to initiate a viscous plume solution.
  - Module was added to output a binary data file(\*plm.vnp) containing the spatial distribution of the flowfield properties that are calculated by the VNAP2 module. This file is in the same format as a RAMP2 binary flowfield

file so that the results can be displayed using the RAMP2 post processing module.

- Miscellaneous updates and modifications were made to monitor program execution, results and error detection.
- The CEC module was modified to produce a data file(\*cec.sav) that contains chamber conditions, transport and thermodynamic properties required for automatic VNAP2 input file generation. CEC module execution for a particular case is not necessary but the user must be able to manually input/change transport(viscosity and conductivity) and thermodynamic data via the graphical user interface. The CEC module automatically generates the correct data. It is recommended that the CEC module be used for generating thermodynamic data for all VNAP2 cases even if there is a gas that can be simulated with constant molecular weight and specific heat ratios(like room temperature air, nitrogen, argon and helium).
- RAMP2 was modified to:
  - Generates data files containing startline properties(\*str.ram) and exit plane lip properties(\*lip.ram) for automatic VNAP2 input file generation. These files are generated during the required RAMP2 nozzle solution.
  - Sets up an exit plane start line using the exit plane data file generated by VNAP2 and subsequently performs a plume solution

The VNAP2 flowfield code is not an actual module of the PEP code. The VNAP2 code is accessed during a PEP execution session via a system call. Upon receiving the system call VNAP2 is executed and the PEP code waits until VNAP2 is finished. Communication of data between PEP and VNAP2 is done via data files that are opened and closed in both codes. The VNAP2 control and input routines are physically a part of the PEP code. The next section provides a step-by-step description on how to generate a viscous high altitude exhaust plume that uses a VNAP2 nozzle flow solution.

## **2.2 Using the VNAP2 module in PEP**

The VNAP2 module is generally used to provide nozzle exit plane properties for calculating high altitude(>300000 feet) exhaust plumes from rocket engines that have relatively low chamber pressures(<50 psia) and high area ratio nozzles(>100). The VNAP2 module is restricted to ideal gas solutions so that exhaust plumes using VNAP2 nozzle data must be calculated using the ideal gas total enthalpy approximation. The VNAP2 input generation module calculates a averaged specific heat ratio and molecular weight that is used throughout the solution. In general this approximation will result in conservative plume induced environments. The following are step-by-step instructions for generating a rocket exhaust plume that includes a viscous nozzle flowfield solution from the VNAP2 module:

- 1) Obtain all motor operating conditions(chamber pressure, propellant and O/F ratio), geometry(preferably including the combustion chamber data) and vehicle operating parameters(altitude, angle of attack and mach number).
- 2) Specify a new problem identification upon accessing the PEP code.
- 3) Input the applications GUI for the problem. This will be automatically be accessed for a new problem.
- 4) Access the CEC module and input the propellant and combustion chamber data via the CEC input generation module. Set up a constant O/F/constant total enthalpy case. The CEC module need not be utilized but the user must already have the proper thermodynamic and transport data available for input to RAMP2 and VNAP2.
- 5) Execute the CEC module to generate the CEC thermodynamic and transport data files(\*u10.ram, \*sav.cec).
- 6) Access the RAMP2 module and generate the RAMP2 input data using the RAMP2 input data generation module. You must respond to the control input module queries that you are going to utilize the VNAP2 module to provide properties for the plume calculation. If you utilized the CEC module to provide thermodynamic and transport data for the VNAP2 input module, select the equilibrium/frozen chemistry option for running RAMP2. The code will automatically switch to the ideal gas variable total enthalpy option for the plume solution. Be sure to respond to input data queries to indicate that you do not want multiple passes through the nozzle and boundary solutions.\*

\* Steps 1 through 6 need not be performed during 1 PEP execution session. Steps 7 through 10 must be performed during a single PEP execution session.

- 7) Access the RAMP2 module and run RAMP2 a first time to generate the RAMP2 nozzle solution.
- 8) Access the VNAP2 module from the top control menu. Select the input generation module to automatically generate the VNAP2 input data file(\*inp.vnp). If the input file already exists several options are available for user to select. A GUI will be displayed(see Figures 1 through 3) that allows the user to change selected input variables. Each of these three GUI's all the user to change selected inputs from the three functional parameter groups: Mesh/Convergence Control, Gas Properties and Geometry/Boundary Conditions. Each of these three GUI's are accessed using the mouse and the parameter button at the top of the GUI. Move the arrow to parameter button, hold down left mouse button and drag the arrow down to the desired parameter group. Then let up on button. Make any desired changes to the available input variables. After all three GUI's have been accessed(if desired) use the mouse to select the use button at the bottom of any one of the GUI's to use the data and exit the

GUI module. Variables that are listed but are shaded cannot be changed and are shown for information only. It is not recommended that none of the variables be changed unless the user is familiar with the VNAP2 code. The values and options which have automatically been specified have been selected to provide the most trouble free execution of the VNAP2 module for typical nozzles that require a fully coupled viscous solution.

ParameterGroups: MeshConvergenceControl

NumberOf X MeshStations/LMAX(11-101):	61
NumberOf Y MeshPoints/MMAX(11-51):	31
MaxNumberTimeSteps/NMAX(1E3-1E5):	10000
CFL TimeStepMultiplier/FDT(0.1-1.3):	0.800
1stTimeStepCFL Multiplier/FDT(0.1-1.0):	0.800
SteadyStateTempConvrgCriteria/TCONV(1E-3-1E2):	0.003

Default Use Close

Figure 1 VNAP2 Mesh/Convergence Control Variables Input/Modification GUI

ParameterGroups: GasProperties

FrozenPrandtlNum/PRANFR(0.1~1.0):	0.000	MolecularViscosityExponent/EMU:	0.500
SpecificHeatRatio/GAMMA(1.0~1.667):	1.323	2ndCoefViscosityMultiplier/CLA:	-8.132E-09
GasConstant/RGAS(1, 1545, lbf-ft/lbm):	766.395	2ndCoefViscosityExponent/ELA:	0.500
ChamberPressure/PT(psia):	44.944	ThermalConductivityMultiplier/CK:	1.969E-03
ChamberTemperature/TT(R):	5000.000	ThermalConductivityExponent/EK:	0.500
MolecularViscosityMultiplier/CMU:	1.221E-08	ExitPlanePressure/PE(psia):	0.017

Default Use Close

Figure 2 VNAP2 Gas Property Variables Input/Modification GUI

ParameterGroups: GeometryBoundaryConditions

FlowDimension/NDIM:	Axisymmetric	SolidWallB.C./NOSLIP:	NoSlipAtWall
InletFlowCondtn/MSUPER:	SubsonicInlet	InitDataSurfType/NID:	SubsonicType
TypeWallGeomy/NGEOM:	GeneralWallOpt1	TypeOfFlow/MLM:	BoundaryLayer
TurbulenceMode/ITM:	MixingLengthModel	XCoordOfInlet/XI(in):	-0.452
ThroatRadiusSq/RSTARS(In^2):	0.004	YCoordOfInlet/YI(in):	0.273
InletPressure/PI(psia):	44.913	YCoordOfThroat/RT(in):	0.064
InletVelocityXUI(ft/sec):	409.170	XCoordEndSolution/XE(in):	2.217
InletVelocityYVI(ft/sec):	0.000	XCoordNozzleExit/XEXIT(in):	2.052
InletDensity/ROI(lbm/ft^3):	0.002	YCoordNozzleExit/YEXIT(in):	0.779

Default Use Close

Figure 3 VNAP2 Geometry/Boundary Variables Input/Modification GUI

- 9) After the input generation/update module has been exited, execute the VNAP2 flowfield generation module. The user will be informed that the module has been entered and finished. There will be intermediate outputs to the screen every 10 time steps to inform the user of the codes progress. The VNAP2 solutions module can take a relatively long time to calculate the nozzle depending on the number of grid points and time steps that are specified(as well as the platform PEP/VNAP2 is being run on). Once the VNAP2 solution has been successfully completed return to the top menu and select the RAMP2 module.
- 10) Execute the RAMP2 solution module for the plume solution. Following the plume solution the RAMP2 post processing module can be accessed to provide plot files that can be used with TECPLOT or PLOT3D to display contour plots of both the nozzle and plume solutions. The VNAP2 binary flowfield file is \*plm.vnp. This file can be post processed from the RAMP2 post processing module by instructing the code that it is a RAMP2 format flowfield type. The plume flowfield(which includes the nozzle inviscid flowfield) is the file \*u3.ram.
- 11) The plume induced environment modules can then be access and the file \*u3.ram may be used to provide flowfield properties for generating the induced environments

Table 1 presents a list files and descriptions that are generated when the VNAP2 module is accessed for a particular problem.

Table 1  
Files Generated and Used During A PEP/VNAP2 Nozzle/Plume Solution

File Name	Description	Generated By	Used By
*sav.cec	Thermodynamic and Transport Property Data	CEC	VNAP2/ RAMP2
*lip.ram	RAMP2 flowfield properties at the nozzle lip	RAMP2	VNAP2 Input Module
*str.ram	RAMP2 properties at startline	RAMP2	VNAP2 Input Module
*inp.vnp	VNAP2 input file	VNAP2 Input Module	VNAP2
*out.vnp	VNAP2 printed output file.	VNAP2	User
*str.vnp	VNAP2 exit plane property distributions	VNAP2	RAMP2
*plm.vnp	VNAP2 binary flowfield property binary data	VNAP2	RAMP2 Post Processing

## 2.3 Description of the VNAP2 Flowfield Module Input Variables

The file \*inp.vnp supplies the input data to the VNAP2 flowfield module of the PEP code. All options that were available in the version of the VNAP2 code that is described in Reference 1 can be run with the version that has been integrated into the PEP code. However, as been discussed above only two types of calculations can be input via the input generation and GUI input modules. Any other options will require the used to manually generate/modify the \*inp.vnp namelist input file prior to running the VNAP2 module. A complete description of the VNAP2 input variables is presented below. The description of the variables were taken almost verbatim from Reference 1. Variables that are shown shaded(**MAIN**) and shaded with an underline(**MAIN**) are variables that are automatically set by the VNAP2 automatic input generator that was developed for the use of VNAP2 with the PEP code to generate a viscous exit plane startline. The underline shaded variables(**MAIN**) can be updated and changed with the PEP input update GUI module. All other variables are the default values that are set in the VNAP2 code. The variable defaults that are listed in the description of input variables that are shown in brackets - (0.0) ,are the original default values that are set in the VNAP2 code. The default values specified above these values are those set by the PEP automatic input file generation module. The problem description record and 10 namelists must be contained in the \*inp.vnp file in the order that they are presented below. Each namelist must be included even if there are no variables contained in the input namelist.

### Problem Description

#### Format:20A4

Column	Parameter	Description
1-80	TITLE	Record containing up to 80 alphanumeric characters that describe the problem. This record must always be the first card in the input file, even if no information is contained on this record.

### Namelist CNTRL

This namelist reads in the parameters that control the overall logic of the program.

Parameter	Description	Default
<b>LMAX</b>	An integer specifying the number of mesh points in the x direction with a maximum value specified by a <b>PARAMETER(101 Max)</b> statement.	61 (None)
<b>MMAX</b>	An integer specifying the number of mesh points in the y direction with a maximum value specified by a <b>PARAMETER(51 Max)</b> statement.	31 (None)
<b>NMAX</b>	An integer specifying the maximum number of time steps. For <b>NMAX=0</b> . Only the initial data surface is computed and printed (provided <b>NPRINT&gt;0</b> ).	10000 (0)
<b>NPRINT</b>	An integer specifying the amount of output desired. For <b>NPRINT=N</b> , every Nth solution plane, plus the initial-data and final solution planes, is printed. For <b>NPRINT=-N</b> , every Nth solution plane, plus the final solution plane, is printed. For <b>NPRINT=0</b> , only the final solution plane is printed.	0
<b>TCNV</b>	Specifies the axial velocity steady-state convergence tolerance in percentage. If equal to zero, the convergence is not checked. This parameter is a function of the problem as well as of grid spacing and, therefore, should be used carefully.	.003 (0.0)
<b>FDT</b>	The parameter A in Eqs.(67)-(69) of Reference 1 that premultiplies the allowable CFL time step. It is desirable to use as large a value of <b>FDT</b> as possible without causing the computation to become unstable. Values as large as 1.3 have been used successfully for shockfree flows, but smaller values are required for flows with shocks (see Sec.II.F of Reference 1 ).	0.8 (0.9)

## Namelist CNTRL(Cont.)

Parameter	Description	Default
<b>FDTI</b>	The same as FDT, except it applies on the first time step only. Because the viscous contribution to the time-step limitation is not used on the first time step, FDTI may be used to get the calculation started with a small time step, without having to use this small value for the entire calculation. Some flows may require a small time step for the first few steps owing to initial gradients in the flow variables. This is often true for viscous flows when the Quick Solution option is used. For this situation, make a short run with small enough values of FDT or FDTI so that the code will run. Then use the restart option (see IPUNCH) to continue the run with more desirable values of FDT or FDTI. For any long running problem, it is usually worth experimenting with FDT and FDTI (as well as VDT and VDTI) to make sure that optimum values are being used.	0.8 (0.9)
<b>FDTI</b>	The same as FDT, except it applies only in the subcycled part of the mesh. That is, FDTI is used from $M = M_{VCB}$ to $M = M_{VCT}$ (see Namelist VCL).	0.8 (1.0)
<b>VDT</b>	The parameter A1 in Eqs. (67)-(69) of Reference 1 that premultiplies the viscous part of the time-step equation, whereas FDT premultiplies the entire time step. Increasing VDT increases the time step.	0.25
<b>VDTI</b>	The same as VDT, except it applies only in the subcycled part of the mesh. That is, VDTI is used from $M = M_{VCB}$ to $M = M_{VCT}$ (see Namelist VCL). The default value is 0.25, although values larger than 1.0 have been used in free-shear layers.	0.25
<b>GAMMA</b>	Specific heat ratio.	1.4
<b>RGAS</b>	Denotes the gas constant in lbf-ft/lbm-deg R if English units are used, or J/kg-deg K if metric units are used.	53.35
<b>TSTOP</b>	Specifies the physical time, in seconds, at which the computations will stopped.	1.0
<b>IUI</b>	An integer specifying the type of units to be used for the input quantities. IF IUI=1, English units are assumed; if IUI=2, metric units are assumed. In using any default values, make sure the values correspond to the proper units.	1
<b>IUO</b>	The same as IUI except for output quantities. IF IUO=3, both English and metric units are printed.	1
<b>IPUNCH</b>	An integer which, if nonzero, punches (writes) the last solution plane on cards (disc or tape) for restart.	0
<b>NPLOT</b>	An integer which, if greater than or equal to zero, plots both velocity vectors and contours of density, pressure, temperature, Mach number, turbulence energy, and dissipation rate on an SC-4020 microfilm recorder. For NPLOT =N, all Nth solution planes, plus the initial-data and final solution plane, are plotted. For NPLOT=0, only the final solution plane is plotted. <b>Note: this option is not operational since the SC-4020 routine calls are dummied out.</b>	-1
LPP1,MPP1 LPP2,MPP2 LPP3,MPP3	Three sets of integers that specify three grid points(the first point is $L=LPP1, M=MPP1$ ) for which the pressure is printed at each time step. When $MPP1(MPP2 \text{ or } MPP3) = M_{DFS}$ not equal 0(Namelist DFSL), the upper dual-flow space wall value is printed. This pressure history is very useful for determining when subsonic flows have reached steady state. If $LPP1 < 0$ , the pressure at each subcycled grid point (see $M_{VCB}$ and $M_{VCT}$ in Namelist VCL) is also printed.	0
<b>NASM</b>	An integer specifying which part of the flow field is tested for steady-state convergence. For NASM=0, the entire flow field is tested. For NASM=1, the transonic and supersonic (throat region to exit) regions are tested.	1

**Namelist CNTRL(Cont.)**

Parameter	Description	Default
NAME	An integer that, when nonzero, causes the 10 namelists to be printed in addition to the regular output.	0
NCONV1	An integer specifying how many times the convergence tolerance TCONV must be satisfied on consecutive time steps before the solution is considered to have converged.	1
IUNIT	An integer that, when equal to zero, causes the program to use either English or metric units (see IUI and IUO). For IUNIT=1, a nondimensional set of units is used.	0
PLOW	If the pressure becomes negative during a calculation, it is set equal to PLOW in psia or kPa.	0.01
ROLOW	If the density becomes negative during a calculation, it is set equal to ROWOL in lb/ft <sup>3</sup> or kg/m <sup>3</sup> .	0.0001
IVPTS	An integer that controls the scaling of the velocity vector plots. IVPTS=1 produces one plot with the maximum vector equal to 0.9 delta x, where delta x is the average value. IVPTS=2 produces the above plot and a second plot where the maximum vector is 1.9 delta x and so on. <b>Note: this option is not operational since the SC-4020 routine calls are dummied out.</b>	1
<u>PRANFR</u>	Frozen Prandtl Number that is not used by VNAP2 module but is passed along from the RAMP2 module solution to the plume solution that used the VNAP2 exit plane results.	1.0

**Namelist IVS**

This namelist specifies the flow variables for the initial-data surface.

Parameter	Description	Default
<u>N1D</u>	<p>An integer specifying the type of initial-data surface desired. For N1D=0, a 2D initial-data surface is read in. A value of U,V,P, and RO (discussed below) must be read in for all mesh points from L=1 to LMAX and from M=1 to MMAX. In addition, for dual-flow-space examples, values of UL,VL, PL and ROL (discussed below) must be read in for all mesh points from L=LDFSS to LDFSF. For the single-equation turbulence model, a value of Q along with QL for the dual-flow-space example, may be read in. For the two-equation model, a value of E, along with EL for the dual-flow-space example, may also be read in. If the arrays Q and QL and the arrays E and EL are not read in, they are set equal to FSQ and FSE (Namelist TURBL), respectively. Values of Q and E may be read in for either N1D=0 or N1D not equal to 0. For N1D not equal to 0, a 1D data surface is computed internally. The following combinations are possible:</p> <p style="text-align: center;"> N1D=-2 subsonic      See RSTAR and RSTARS  N1D=-1 subsonic      See RSTAR and RSTARS  N1D=1 subsonic-sonic-supersonic      No  N1D=2 subsonic-sonic-subsonic      additional  N1D=3 supersonic-sonic-supersonic      data are  N1D=4 supersonic-sonic-subsonic      needed </p>	1

**Namelist IVS(Cont.)**

<b>Parameter</b>	<b>Description</b>	<b>Default</b>
U(L,M,1)	An array denoting the x-direction velocity components in ft/s or m/s. For N1D=0, U(L,M,1) must be read in for cases from L=1 to LMAX and from M=1 to MMAX. For N1D not equal to 0, U(L,M,1) is not read in.	None
V(L,M,1)	An array denoting the y-direction velocity component in ft/s or m/s. See U(L,M,1) for additional information.	None
P(L,M,1)	An array denoting the pressure in psia or kPa. See U(L,M,1) for additional information.	None
RO(L,M,1)	An array denoting the density in lbm/ft <sup>3</sup> or kg/m <sup>3</sup> . See U(L,M,1) for additional information.	None
Q(L,M,1)	An array denoting the turbulence energy in ft <sup>2</sup> /s <sup>2</sup> or m <sup>2</sup> /s <sup>2</sup> . See U(L,M,1) for additional information. The default value is FSQ(M) in Namelist TURBL	FSQ(M)
E(L,M,1)	An array denoting the dissipation rate in ft <sup>2</sup> /s <sup>3</sup> or m <sup>2</sup> /s <sup>3</sup> . See U(L,M,1) for additional information. The default value is FSE(M) in namelist TURBL	FSE(M)
UL(L,1)	An array denoting the x-direction velocity component in ft/s or m/s and corresponding to the lower dual-flow-space wall. The values for the upper dual-flow-space wall are read in by U(L,MDFS,1). For N1D=0 and MDFS not equal to 0, UL(L,1) must be read in for cases from L=LDFS to LDFS. For N1D not equal to 0 or MDFS=0, UL(L,1) is not read in.	None
VL(L,1)	An array denoting the y-direction velocity component in ft/s or m/s. See UL(L,1) for additional information.	None
PL(L,1)	An array denoting the pressure in psia or kPa. See UL(L,1) for additional information.	None
ROL(L,1)	An array denoting the density in lbm/ft <sup>3</sup> or kg/m <sup>3</sup> . See UL(L,1) for additional information.	None
QL(L,1)	An array denoting the turbulence energy in ft <sup>2</sup> /s <sup>2</sup> or m <sup>2</sup> /s <sup>2</sup> . See UL(L,1) for additional information. The default value is FSQ(L) in Namelist TURBL	FSQ(L)
EL(L,1)	An array denoting the dissipation rate in ft <sup>2</sup> /s <sup>3</sup> or m <sup>2</sup> /s <sup>3</sup> . See UL(L,1) for additional information. The default value is FSEL in Namelist TURBL	FSEL
<b>RSTAR RSTARS</b>	If N1D=-1 or -2, either RSTAR for planar or RSTARS for axisymmetric flow must be read in. RSTAR is the area per unit depth or height (in in. or cm) where the Mach number is unity. RSTARS is the area divided by pi that is the radius squared (in in. <sup>2</sup> or cm <sup>2</sup> ) where the Mach number is unity.	None

**Notes on Namelist IVS:**

If the restart option is to be used, the initial run must be made with IPUNCH not equal 0 in CNTRL, thereby causing a new IVS Namelist deck to be written on disc a file. The new IVS Namelist replaces the one used initially and includes two additional parameters, NSTART and TSTART, which denotes, respectively, the time step and the physical time where the solution was restarted.

When N1D is not equal to 0, the initial data are calculated using 1D isentropic theory. However, the x and y velocity components are adjusted while the magnitude is kept constant and the flow angle is satisfied. The flow angles are linearly interpolated between the slope of the wall and the centerbody. For the dual-flow-space example, the Mach number is assumed to be equal in both flow spaces at a given value of x. However, the flow angles are interpolated between the centerbody and the lower dual-flow-space boundary for the lower space and between the upper dual-flow-space boundary and the wall for the upper space.

**Namelist GEMTRY**

This namelist specifies the parameters that define the wall contour.

Parameter	Description	Default
<b>NDIM</b>	An integer denoting the flow geometry. For NDIM=0, 2D planar flow is assumed, and for NDIM=1, axisymmetric flow is assumed.	1
<b>NGEOM</b>	An integer specifying one of four different wall geometry's. A discussion of these four cases follows the definitions of the additional parameters in this namelist.	3 (none)
<b>XI</b>	The x coordinate, in in. or cm, of the wall inlet.	None
<b>RI</b>	The y coordinate, in in. or cm, of the wall inlet.	None
<b>RT</b>	The y coordinate, in in. or cm, of the wall throat.	None
<b>XE</b>	The x coordinate, in in. or cm, of the last solution station(wall or free-jet exit).	None
<b>RCI</b>	The radius of curvature, in in. or cm, of the wall inlet.	None
<b>RCT</b>	The radius of curvature, in in. or cm, of the wall throat.	None
<b>ANGI</b>	The angle, in degrees, of the converging section.	None
<b>ANGE</b>	The angle, in degrees, of the diverging section.	None
<b>XWI</b>	A 1D array of non-equally spaced x coordinates in in. or cm.	None
<b>YWI</b>	A 1D array of y coordinates, in in. or cm, corresponding to the x coordinates in array XWI.	None
<b>NWPTS</b>	An integer specifying the number of entries in arrays XWI and YWI. The maximum value is specified by a PARAMETER(101 max) statement (see Sec. II.E.1 of Reference 1).	None
<b>IINT</b>	An integer specifying the order of interpolation used. The maximum value is 2	2
<b>IDIF</b>	An integer specifying the order of differentiation used. The maximum value is 5.	2
<b>YW</b>	A 1D array of y coordinates, in in. or cm, which correspond to LMAX x coordinates given by XP in Namelist VCL.	None
<b>NXNY</b>	A 1D array (floating point) of the negative of the wall slopes corresponding to the elements of YW.	None
<b>JFLAG</b>	An integer that, when equal to 1, denotes that a free-jet calculation is to be carried out and, when equal to -1, denotes that a supersonic sharp expansion corner is present on the wall. These two options are allowed only for the free-slip wall boundary condition. Many free-jet flows contain shocks and will therefore, require artificial viscosity (see Namelist AVL).	0
<b>LJET</b>	An integer that, when JFLAG=1, denotes the first mesh point of the free-jet boundary (the last wall mesh point is LJET-1. However, when JFLAG=-1, LJET is the next mesh point downstream of the sharp expansion corner (the corner mesh point is LJET-1). The program assumes that either the wall ends exactly at LJET-1 (JFLAG=1) or the sharp expansion corner is located exactly at LJET - 1( JFLAG = -1). Also, for the sharp expansion corner case (JFLAG=-1), the slope of the wall at the corner (LJET-1) should be the upstream value. The program does not allow both a sharp expansion corner and a free-jet calculation. In addition LJET must be > 2 and < LMAX - 1.	None
<b>XEXIT</b>	Axial location of the nozzle exit in in. or cm. This variable is determined from the RAMP2 module inputs and passed onto VNAP2 for determining exit plane properties that are passed on to RAMP2 for the plume solution.	None
<b>REXIT</b>	Radial location of the nozzle lip of in in. or cm. This variable is determined from the RAMP2 module inputs and passed onto VNAP2 for determining exit plane properties that are passed on to RAMP2 for the plume solution.	None

### Notes on VNAP2 geometry input options:

The default option that is used by the AI code in setting up the geometry for using the VNAP2 code to establish exit plane flow properties is NGEOM=3. The user cannot enter another option using the built in menu. The following is a discussion of the four different wall geometry's considered by this program:

- a. Constant Area Duct (NGEOM=1). The parameters XI,RJ(radius of the duct) and XE must be specified.
- b. Circular-Arc, Conical Wall (NGEOM=2). The geometry for this case is shown in Fig. 21 of Reference 1. The parameters XI, RI, RT, XE, RCI,RCT, ANGI, and ANGE are specified. The x coordinate of the throat and the radius of the exit are computed internally.
- c. General Wall (NGEOM=3). An arbitrary wall contour is specified by tabular input. NWPTS x- and y-coordinates pairs are specified by the arrays XWI and YWI, respectively. The tabular data need not be equally spaced. From the specified values of NWPTS, XWI, YWI, IINT, and IDIF, the program uses IINT-order interpolation to obtain LMAX y coordinates that correspond to the x coordinates given by XP in Namelist VCL. Next, IDIF-order differentiation is used to obtain the wall slope at these LMAX points.
- d. General Wall (NGEOM=4). An arbitrary wall contour is specified by tabular input. LMAX y coordinates and the negative of their slopes are specified by the arrays YW and NXNY, respectively. These y coordinates correspond to the LMAX x coordinates given by XP in Namelist VCL, XI and XE also must be read in.

### Namelist GCBL

This namelist specifies the parameters that define the centerbody geometry. If no centerbody is present, this namelist is left blank but must still be present in the data deck. None of the variables specified by this namelist are used by the options necessary to generate nozzle exit plane flow properties for the PEP code so that a blank namelist is input.

Parameter	Description	Default
NGCB	An integer that, when nonzero, specifies one of four different centerbody geometry's. A discussion of these four cases will follow the definitions of the additional parameters in this namelist.	0
RICB	The y coordinate, in in. or cm, of the centerbody inlet.	None
RTCB	The y coordinate, in in. or cm, of the centerbody maximum radius.	None
RCICB	The radius of curvature, in in. or cm, of the centerbody inlet.	None
RCTCB	The radius of curvature, in in. or cm, of the centerbody maximum radius.	None
ANGICB	The angle, in degrees, of the converging section.	None
ANGECEB	The angle, in degrees, of the diverging section.	None
XCBI	A 1D array of non-equally spaced x coordinates in in. or cm.	None
YCBI	A 1D array of y coordinates, in in. or cm, corresponding to the x coordinates in array XCBI.	None
NCBPTS	An integer specifying the number of entries in arrays XCBI. The maximum value is specified by a PARAMETER(101 max) statement (see Sec. II.E.1 of Reference 1).	None
IINTCB	An integer specifying the order of interpolation used. The maximum value is 2.	2
IDIFCB	An integer specifying the order of differentiation used. The maximum value is 5.	2
YCB	A 1D array of y coordinates, in in. or cm, which correspond to LMAX x coordinates given by XP in Namelist VCL.	0.0
NXNYCB	The 1D array (floating point) of the negative of the centerbody slopes corresponding to the elements of YCB.	0.0

The following is a discussion of the four different centerbody geometry's considered by the VNAP2 program that can be input in the GBCL namelist:

- a. Cylindrical Centerbody (NGCB=2). The parameter RICB (radius of the centerbody) must be specified.
- b. Circular-Arc, Conical Centerbody (NGCB=2). The geometry for this case is shown in Fig. 22 of Reference 1. The parameters RICB, RTCB, RCICB, RCTCB, ANGICB and ANGECEB are specified. The x coordinate of the maximum radius and the radius of the exit are computed internally.
- c. General Centerbody (NGCB=3). An arbitrary centerbody contour is specified by tabular input. NCBPTS x- and y-coordinate pairs are specified by the arrays XCBI and YCBI, respectively. The tabular data need not be equally spaced. From the specified values of NCBPTS, XCBI, YCBI, IINTCB, and IDIFCB, the program uses IINTCB-order interpolation to obtain LMAX y coordinates that correspond to the x coordinates given by XP in Namelist VCL. Next, IDIFCB-order differentiation is used to obtain the centerbody slope at these LMAX points.
- d. General Centerbody (NGCB=4). An arbitrary centerbody contour is specified by tabular input. LMAX y coordinates and the negatives of their slopes are specified by the arrays YCB and NXNYCB, respectively. These y coordinates correspond to the LMAX x coordinates given by XP in Namelist VCL.

#### Namelist BC

This namelist specifies the flow boundary conditions for all computational boundaries.

Parameter	Description	Default
NSTAG	An integer that, when nonzero, denotes that variable total pressure PT, variable total temperature TT, and variable flow angle THETA (all discussed below) have been specified. If NSTAG not equal to 0, then a value for PT, TT, and THETA must be specified at all the points from M=1 to MMAX, even if one or two of the variables are constant or some grid points are not used (ISUPER=2 or 3). If NSTAG=0, only the first value for each of the three arrays needs to be specified.	0
<u>PT(M)</u>	A 1D array denoting the stagnation pressure, in psia or kPa, across the inlet (see ISUPER). This array is used to calculate the 1D initial-data surface as well as the inflow conditions for ISUPER=0, 2, or 3.	None
<u>TT(M)</u>	A 1D array denoting the stagnation temperature in degrees R or K, across the inlet (see ISUPER). This array is used to calculate the 1D initial-data surface as well as the inflow conditions for ISUPER=0, 2, or 3.	None
THETA(M)	A 1D array denoting the flow angle, in degrees, across the inlet (see ISUPER). The default value is THETA(1)=0.0, which is meaningful only when NSTAG=0.	0.0
PTL	Denotes the stagnation pressure, in psia or kPa, at the point where the lower dual-flow-space wall intersects the inlet (see Namelist DSFL). The upper dual-flow-space wall value is read in by PT(MDFS). If NSTAG=0 or MDFS=0 or LDFS not equal to 1, then PTL is not read in.	None
TTL	The same as PTL, except denotes the stagnation temperature in degrees R or K	None
THETAL	The same as PTL, except denotes the flow angle is degrees.	None

## Namelist BC(Cont.)

Parameter	Description	Default
PE(M)	A 1D array denoting the pressure, in psia or kPa, to which the flow is exiting. This pressure is used to compute the flow exit conditions when the flow is subsonic, the free-jet boundary location when a free-jet calculation is requested, or the wall inflow-outflow boundary when IWALL=1. The free-jet or wall inflow/outflow boundary pressure is assumed to be constant and equal to PE(MMAX). Subroutine WALL could be modified to allow PE to be a function of x or t. This array starts with the centerline or centerbody value and ends with the wall value. If the exit pressure is constant, only the first value of the array needs to be read in.	Exit pressure from RAMP2 (14.7)
PEL	Denotes the pressure, in psia or kPa, to which the flow is exiting at the point where the lower dual-flow-space wall intersects the exit (see Namelist DFSL). The upper dual-flow-space wall value is read in by PE(MDSF). If MSSF=0 or LDFSE=LMAX, PEL is not read in.	None
UI(M)	A 1D array denoting the x velocity, in ft/s or m/s, across the inlet (see ISUPER). This array, as well as the arrays, VI, PI, and ROI below, starts with the centerline or centerbody value and ends with the wall value. Values must be specified for points from M=1 to MMAX even if some grid points are not used (ISUPER=2 or 3).	None
VI(M)	The same as UI, except y velocity.	None
PI(M)	The same as UI, except denotes pressure in psia or kPa.	None
ROI(M)	The same as UI, except denotes density in lbm/ft <sup>3</sup> or kg/m <sup>3</sup> .	None
UIL	Denotes the x velocity in ft/s or m/s at the point where the lower dual-flow-space wall intersects the inlet (see Namelist DFSL). The upper dual-flow-space wall value is read in by UI(MDSF). For MDFS=0 or LDFSS not equal to 1, UIL is not read in. See ISUPER for additional information.	None
VIL	The same as UIL, except y velocity.	None
PIL	The same as UIL, except denotes pressure in psia or kPa.	None
ROIL	The same as UIL, except denotes density in lbm/ft <sup>3</sup> or kg/m <sup>3</sup> .	None
TW	A 1D array denoting the wall temperature in degrees R or K corresponding to the x mesh points. If TW is not specified, the wall is assumed to be adiabatic.	None
TCB	The same as TW, except denotes centerbody temperature.	None
TL	The same as TW, except denotes lower dual-flow-space wall (see Namelist DSFL). If MDFS=0, TL is not read in.	None
TU	The same as TW, except denotes upper dual-flow-space wall (see Namelist DSFL). If MDFS=0, TU is not read in.	None
ISUPER	An integer that specifies whether the inlet flow is subsonic, supersonic, or both ISUPER may have the following values:  ISUPER=0 Subsonic inflow with PT,TT, and THETA as the specified quantities. ISUPER=1 Subsonic, supersonic or mixed inflow with UI,VI,PI, and ROI as the specified quantities. For subsonic flow, PI is only an initial guess if INBC=0, and UI is only an initial guess if INBC not equal to 0. ISUPER=2 Subsonic, supersonic, or mixed inflow between the centerbody and lower dual-flow-space wall with UI,VI, PI, and ROI as the specified quantities. For subsonic flow, PI is only an initial guess if INBC=0, and UI is an initial guess if INBC not equal to 0.	0

## Namelist BC(Cont.)

Parameter	Description	Default
ISUPER	<p>ISUPER=2 is subsonic inflow between the upper dual-flow-space wall and the wall with PT,TT, and THETA as the specified quantities.</p> <p>ISUPER=3 The same as ISUPER=2, except subsonic and supersonic, supersonic or mixed sides are switched.</p> <p><b>Note: For nozzle cases where the PEP code has geometry for the combustion chamber ISUPER is set to 0. For cases where PEP is solving the flowfield beginning from the throat ISUPER is set to 1.</b></p>	
INBC	An integer that specifies whether u or p will be the inflow boundary condition for ISUPER not equal to 0. If INBC=0, u is the boundary condition and p is calculated. If INBC not equal to 0, the reverse is true.	0
IWALL	<p>An integer that denotes whether the wall is a solid boundary(includes free-jet option) or a constant pressure inflow/outflow boundary that is fixed with respect to time.</p> <p>IWALL=0 Specifies a solid or free-jet boundary.</p> <p>IWALL=1 Specifies a constant pressure PE(MMAX) boundary. When there is inflow across this constant pressure boundary, u and p are set equal to the wall-inlet value. This option cannot be used with JFLAG not equal to 0 in Namelist GEMTRY.</p>	0
IWALLO	An integer that, when not equal to 0, forces linear extrapolation of the pressure at the wall for the IWALL=1 case. This option is useful when a shock wave exits the wall boundary or when the flow normal to the boundary is supersonic outflow.	0
IINLET	An integer that, when not equal to 0, forces specification of all variables as the inflow boundary condition regardless of the Mach number. It applies only when ISUPER not equal to 0.	0
IEXITT	An integer that, when not equal to 0, forces either extrapolation (IEXITT=1) or specified pressure (IEXITT=2) as the outflow boundary condition regardless of the Mach number.	0
IEX	An integer that denotes the type of extrapolation to be used for supersonic outflow. IEX=0 denotes zeroth-order extrapolation, and IEX=1 denotes linear extrapolation.	1
IVBC	An integer that specifies whether extrapolation or reflection is used to determine the viscous terms at boundaries. IVBC=0 specifies reflection, IVBC=1 specifies linear extrapolation, and IVBC=2 specifies zeroth-order extrapolation. Reflection is always used at the centerline or midplane. The adiabatic wall boundary condition (that is TW,TCB,TL and TU not specified) requires IVBC=0.	0
<u>NOSLIP</u>	An integer that, when equal to zero, specifies free-slip walls whereas NOSLIP=1 specifies no-slip (u=v=0) walls for all solid boundaries. The no-slip boundary condition is not enforced at the wall when IWALL=0.	1 (0)
DYW	A parameter that specifies the maximum change that is allowed on each time step in the free-jet boundary location. The default value is 0.001, that is 0.1% maximum change per time step.	.001

**Namelist BC(Cont.)**

Parameter	Description	Default
IAS	An integer that, if not equal to zero, causes the upper and lower dual-flow-space wall slopes to be set equal to the average of the two slopes. This occurs only at the point or points where the two dual-flow-space walls intersect. That is, for LDFSS not equal to 1, the slopes at LDFSS will be set equal on their average. Also, if LDFSF not equal to LMAX, the same occurs.	0
ALI	The coefficient $C_a$ is Eqs. (55) and (56) of Reference 1. This coefficient controls the nonreflecting inflow boundary condition employed at the left boundary. Any nonzero value will activate the nonreflecting option; however, values of approximately 0.1 appear to work well for many problems. Specifying ALI not equal to 0.0 for the $P_t$ , $T_t$ , and flow angle boundary condition or supersonic inflow has no effect.	0.0
ALE	The coefficient $C_a$ in Eq. (54) of Reference 1. This coefficient controls the nonreflecting inflow and outflow boundary condition at the right boundary. See ALI for further details. Specifying ALE not equal to 0.0 for supersonic outflow has no effect.	0.0
ALW	The coefficient $C_a$ in Eq. (54) of Reference 1. This coefficient controls the nonreflecting inflow and outflow boundary condition at the wall boundary. See ALI for further details. Specifying ALW not equal to 0 when IWALL=0 (Namelist BC) has no effect.	0.0

**Namelist AVL**

This namelist specifies the parameters that determine the artificial viscosity used to stabilize the calculations for shocks and control space-and time-smoothing option. For flows without shocks or where space or time smoothing is not desired, this namelist is left blank. See Sec. II.F of Reference 1 for additional information. None of the variables specified by this namelist are used by the options necessary to generate nozzle exit plane flow properties for the PEP code so that a blank namelist is input.

Parameter	Description	Default
CAV	Denotes the artificial viscosity premultiplier $C$ in Eq. (23) of Reference 1. Sec. II.F of Reference 1 for typical values.	0.0
XMU	Denotes the coefficient $C_{u1}$ in Eq. (24) of Reference 1 in the artificial viscosity model. A non-dimensional value is used.	0.4
XLA	Denotes the coefficient $C_{\lambda}$ in Eq. (23) of Reference 1 in the artificial viscosity model. A nondimensional value is used.	1.0
PRA	Denotes the coefficient $Pr_A$ in Eq. (25) of Reference 1 in the artificial viscosity model and represents an artificial Prandtl number.	0.7
XRO	Denotes the coefficient $C_{r\theta}$ in Eq. (26) of Reference 1 in the artificial viscosity model.	0.6
LSS, LSF	Integers that specify the x mesh points at which the addition of the artificial viscosity will begin (LSS) and end (LSF). These parameters can significantly reduce the run time for inviscid flows where a shock occupies only a small part of the flow.	1 999
MSS, MSF	The same as LSS and LSF, except that these specify the y mesh points at which the addition of the artificial viscosity begins (MSS) and ends (MSF).	1 999

## Namelist AVL(Cont.)

Parameter	Description	Default
IDIVC	An integer that, when not equal to 0, bypasses the check on the sign of the velocity divergence in the artificial viscosity model. That is, the artificial viscosity will be nonzero for both expansions and compression's. This improves some complex multiple shock interactions, but also increases the smearing of expansion.	0
ISS	An integer that, when not equal to 0, adds the sound speed gradient to the velocity divergence in Eq. (23) of Reference 1. For ISS=1, the sound speed gradient is added to the velocity divergence only if the velocity divergence is < 0. For ISS=2, the sound speed gradient is always added. This term improves contact surface calculations (see Sec. I.F of Reference 1.)	0
SMACH	Denotes the Mach number below which no artificial viscosity for shock calculations is added to the solution. This option is useful for moderate-to-high Reynolds number, steady flow, where the artificial viscosity swamps the molecular and turbulent viscosity's in the boundary layer. By setting SMACH equal to ~ 0.5, the artificial viscosity is zero for most of the subsonic part of the boundary layer. See I.F of Reference 1 for additional details.	0.0
NST	An integer denoting the time step at which a small amount of numerical space or time smoothing is stopped. Smoothing is employed on the regular time steps and not the subcycled steps (see Namelist VCL). This smoothing may be required to stabilize the calculations for very nonuniform or impulsively started initial-data surfaces. Some initial smoothing in space causes subsonic flows to reach steady state faster, but this is not the case for transonic and supersonic flows. Time smoothing also causes subsonic flows to converge to steady state faster. When using the restart option, make sure NST is set equal to zero unless additional smoothing is desired. If additional smoothing is desired on a restart, make sure that the values of SMP or SMPT on the restart equal the final values of the previous run (see SMP and SMPT discussion below).	0
SMP	<p>A Parameter that, along with NST and SMPF, controls the amount of space smoothing (provided NST not equal to 0. SMP must be between 0.0 and 1.0. The dependent variables are smoothed by the following formula:</p> $u_{L,M} = SMP * u_{L,M} + (1.0 - SMP) * (u_{L+1,M} + u_{L,M+1} + u_{L-1,M} + u_{L,M-1}) / 4.0$ <p>The value of SMP changes on each time step by the following replacement formula:</p> $SMP = SMP + (SMPF - \underline{SMP}) / NST,$ <p>where the underlined SMP denotes the original input value. The inlet (L=1) and exit (L=LMAX) columns of grid points are not smoothed.</p>	1.0
SMPF	A parameter that, along with NST and SMP, controls the amount of space smoothing (see SMP for details). SMPF must be between 0.0 and 1.0.	1.0

## Namelist AVL(Cont.)

Parameter	Description	Default
SMPT	<p>A parameter that, along with NST and SMPTF, controls the amount of time smoothing or relaxation (provided NST not equal to 0). The dependent variables are smoothed by the following formula;</p> $u_{L,M}^{N+1} = SMPT * u_{L,M}^{N+1} + (1.0 - SMPT) * u_{L,M}^N$ <p>The value of SMPT changes on each time step by the following replacement formula:</p> $SMPT = SMPT + (SMPTF - SMPT) / NST,$ <p>where the underlined SMPT denotes the original input value. Where some initial space smoothing followed by longer duration time smoothing is desired, flows can be computed using the restart option.</p>	1.0
SMPTF	A parameter that, along with NST and SMPT, controls the amount of time smoothing (see SMPT for details).	1.0
NTST	<p>An integer that specifies the interval of time steps over which the solution is time smoothed (provided NST not equal to 0 and SMART not equal to 1.0. For example, if NTST=10, then after every 10 time steps the solution at the current time step N is time averaged with the solution at time step N - 10. This averaged solution is then stored and used to average with the solution at N + 10. For NTST=0, the code monitors the pressure at the L=LPP1 and M=MPP1 grid point (Namelist CNTRL) and time smoothes when this pressure changes direction. If LPP1 and MPP1 are not specified and NTST=0, there is no time smoothing. This extended-intended-interval time smoothing usually improves the convergence to steady state of subsonic flows. To use this option with NTST=0 or &gt;1, the arrays US, VS, PS, ROS, QS and ES must be dimensioned for LMAX and MMAX, while arrays ULS, VLS, PLS, ROLS, QLS, and ELS must be dimensioned for LMAX. These arrays are located in Common AV.</p>	1
IAV	An integer that, when not equal to 0, causes the viscous-turbulence terms, turbulence energy, and dissipation rate (or length scale) to be printed at the solution planes specified by NPRINT. IAV=2 causes the viscous terms for each subcycled time step to be printed (provided MVCB and MVCT in Namelist VCT are nonzero).	0

## Namelist RVL

This namelist specifies the real or molecular viscosity parameters. For inviscid flows, this namelist is left blank.

Parameter	Description	Default
<u>CMU</u> , <u>EMU</u>	<p>These parameters specify the second molecular viscosity MU by the following equation:</p> $MU = CMU * T^{EMU},$ <p>where T is the temperature in degrees R or K. The units of MU are lbf-s/ft<sup>2</sup> or Pa-s.</p>	.165E-07 0.5

**Namelist RVL(Cont.)**

Parameter	Description	Default
<u>CLA</u> , <u>ELA</u>	These parameters specify the second coefficient of viscosity LAMBDA by the following equation: $\text{LAMBDA} = \text{CLA} * T^{\text{ELA}},$ where T is the temperature in degrees R or K. The units of LAMBDA are lbf-s/ft <sup>2</sup> or Pa-s.	-.11E-07 0.5
<u>CK</u> , <u>EK</u>	These parameters specify the thermal conductivity k by the following equation: $k = \text{CK} * T^{\text{EK}},$ where T is temperature in degrees R or K. The units of k are lbf/s-R or W/m-K.	.143E-03 0.5

**Namelist TURBL**

This namelist specifies the turbulence model parameters. For laminar as well as inviscid flow, Namelist TURBL cannot be blank

Parameter	Description	Default
ITM	An integer that, when nonzero, specifies one of three different turbulence models. ITM=1 specifies a mixing-length model; ITM=2 specifies a one-equation, turbulence energy model; and ITM=3 specifies a two-equation, turbulence energy dissipation-rate model.	1 (0)
IMLM	An integer, required for ITM=1 or 2, that specifies whether the flow is a free shear layer (IMLM=1) or a boundary-layer flow (IMLM=2). This information is required because the equations for the mixing length (ITM=1) and the length scale of the one-equation model (ITM=2) are different depending on whether the flow is a free shear or boundary layer. For single-flow spaces, the shear layer option assumes either that the boundaries are free slip or that the lower boundary is a symmetry boundary and the wall must be a constant pressure inflow/outflow boundary. The boundary-layer option assumes one no-slip boundary, which is either a centerbody or a wall, but not both. For dual-flow spaces (see Namelist DFSL), the dual-flow space walls are assumed to be no-slip boundaries, but the lower boundary must be a symmetry boundary and the wall must be a constant pressure inflow/outflow boundary. The program then uses the boundary-layer option between the dual-flow-space walls and the shear-layer option elsewhere, regardless of IMLM. Therefore, for dual-flow spaces IMLM does not need to be specified.	2 (1)
CML1, CML2	These coefficients, defined in Eqs.(9) and (10) of Reference 1 and required for ITM=1 or 2 or used in the shear-layer option (for IMLM=1 or for dual-flow spaces). The mixing length used in both ITM=1 and 2, is calculated by multiplying shear-layer thickness by these coefficients. CML2 is for velocity profiles where the minimum velocity is in the flow interior, and CML1 is for monatomic profiles. The default values for both coefficients are 0.125 for planar flows and 0.11 for axisymmetric flow.	.125, .11 .125, .11
CAL	Denotes the coefficient alpha bar in the governing equations, Eqs.(1)-(4) of Reference 1. This coefficient controls the effect of variable density for all three turbulence models.	1.0

## Namelist TURBL(Cont.)

Parameter	Description	Default
CQL	This coefficient, which is $C_q$ in Eq.(15) of Reference 1 and required by ITM=2, is multiplied by the mixing length to obtain the length scale used in the one-equation model. The default value is 17.2 for planar flows and 12.3 for axisymmetric flow.	17.2 12.3
CQMU	This coefficient, which is $C_u$ in Eqs(17) and (21) of Reference 1 and required by ITM=2 or 3, pre-multiplies the expression for the turbulent viscosity in the one-and two-equation models.	0.09
C1, C2, SIGQ, SIGE	Coefficients, which are $C_1$ , $C_2$ , $SIGMA_q$ and $SIGMA_e$ , respectively in Eq.(20) of Reference 1 and required by ITM=3, for the two-equation, turbulence energy-dissipation-rate model.	1.44 1.8 1.0 1.3
BFST	A parameter, required by ITM=3, that sets a lower bound for $q$ and $e$ in the two-equation model by the following relation:  $q_{L,M} \text{ G.E. } BFST * FSQ(M),$ $e_{L,M} \text{ G.E. } BFST * FSE(M)$ where FSQ and FSE are defined below. A value between 0.0 and 1.0 is necessary for some separated flows. If MDfs not equal to 0 and $L < LDFSS$ or $L > LDFSF$ (Namelist DFSL), then BFST is set to zero.	0.0
FSQ(M)	A 1D array that denotes the inlet or free-stream turbulence energy level (ITM=2 or 3) in $ft^2/s^2$ or $m^2/s^2$ . This array, as well as the array FSE, starts with the centerline or centerbody value and ends with the wall value.	0.0001
FSE(M)	The same as FSQ, except that the dissipation rate level (ITM=3) is given in $ft^2/s^3$ or $m^2/s^3$ .	0.1
FSQL	Denotes the inlet or free-stream turbulence energy level (ITM=2 or 3) in $ft^2/s^2$ or $m^2/s^2$ at the point where the lower dual-flow-space wall intersects the inlet.(see Namelist DFSL). The upper dual-flow-space wall is read in by FSQ(MDFS). For MDfs=0 or LDFSS not equal to 1, FSQL is not read in.	0.0001
FSEL	The same as FSQL, except that the dissipation rate level (ITM=3) is given in $ft^2/s^3$ or $m^2/s^3$ .	0.1
QLOW	If during a calculation the turbulence energy (ITM= 2 or 3) becomes less than or equal to QLOW, it is set equal to QLOW.	0.0001
ELOW	The same as QLOW except for the dissipation rate (ITM=3).	0.1
LPRINT, MPRINT	Integers that, when greater than zero, cause the convection, production, dissipation, and diffusion terms of the turbulence energy (ITM=2 or 3) and dissipation rate (ITM=3) to be printed for $L = LPRINT$ , $M = MPRINT$ at every time step. The axisymmetric terms are not included.	0 0
PRT	Denotes the turbulent Prandtl number in Eq. (8) of Reference 1. The turbulent viscosity ( $\mu_t$ ) is calculated by the turbulent model, after which the turbulent conductivity ( $k_t$ ) is calculated from PRT.	0.9
STBQ, STBE	Denote the coefficients $C_q$ and $C_e$ , respectively, in Eq. (22) of Reference 1. These coefficients control the fourth-order smoothing for the two-equation model (ITM=3). This smoothing may improve the results for strongly separated flows.	0.0 0.0

### Namelist DFSL

This namelist specifies the dual-flow-space walls. For single-flow-space examples, this namelist is left blank. None of the variables specified by this namelist are used by the options necessary to generate nozzle exit plane flow properties for the PEP code so that a blank namelist is input.

Parameter	Description	Default
MDFS	An integer that, when nonzero, specifies the M row of grid points along which the dual-flow-space walls are positioned. MDFS cannot be set equal to 2 or MMAX-1.	0
LDFSS, LDFSF	Integers that specify the x grid points where the dual-flow-space walls start and end, respectively. LDFSS and LDFSF cannot be set equal to 2 or LMAX - 1, respectively.	0 0
NDFS	An integer specifying one of two different dual-flow-space wall geometry's. A discussion of these two cases follows the definitions of the additional parameters in the namelist.	None
YU, YL	1D arrays of y coordinates in in. or cm, which correspond to the LMAX x coordinates given by XP in Namelist VCL. YU denotes the upper dual-flow-space wall and YL denotes the lower.	0.0 0.0
NXNYU, NXNYL	1D arrays (floating point) of the negative of the dual-flow-space wall slopes corresponding to the elements of YU and YL, respectively.	0.0 0.0
XUI, XLI	1D arrays of non-equally spaced x coordinates in in. or cm. XLI corresponds to the upper dual-flow-space wall and XLI corresponds to the lower.	None None
YUI, YLI	1D arrays of y coordinates in in. or cm, corresponding to the x coordinates in arrays XUI and XLI, respectively.	None None
NUPTS, NLPTS	Integers specifying the number of entries in arrays XUI-YUI and XLI-YLI, respectively. The maximum value is specified by a PARAMETER statement (see Sec. II.E.1 of Reference 1)	None None
IINTDFS	An integer specifying the order of interpolation used. The maximum value is 2.	2
IDIFDFS	An integer specifying the order of differentiation used. The maximum value is 5.	2

The following is a discussion of the two different dual-flow-space wall geometry's considered by VNAP2. If the dual-flow-space walls begin in the interior (LDFSS not equal to 1) the values of YL and YU (or YLI and YUI) for  $L = LDFSS$  must be equal. The same is true at  $L = LDFSF$  if the dual-flow-space walls end in the interior ( $LDFSF = LMAX$ ). If the dual-flow-space walls begin and end in the interior, then the ratio  $(YL - YCB)/(YU - YCB)$  at  $L = LDFSS$  must equal that at  $L = LDFSF$ . The angle of attack of the dual-flow-space walls can be varied somewhat by changing the shape of centerbody and wall. However, if the centerbody and wall shapes are fixed, then the angle of attack cannot be varied.

a. General Dual-Flow-Space Wall ( $NDFS=1$ ). An arbitrary dual-flow-space wall contour is specified by tabular input. NUPTS x and y coordinate pairs are specified the arrays XUI and YUI, respectively, NLPTS x and y coordinate pairs are specified by the arrays XLI and YLI, respectively. The tabular data need not be equally spaced. From the specified values of NUPTS, XUI, YUI, NLPTS, XLI, YLI, IINTDFS, and IDIFDFS, the program uses IINTDFS-order interpolation to obtain  $(LDFSF-LDFSS+1)$  upper and lower dual-flow space wall y coordinates that correspond to the  $(LDFSF - LDFSS + 1)$  x coordinates given by  $XP(LDFSS)$  to  $XP(LDFSF)$  in Namelist VCL. Next, IDIFDFS-order differentiation is used to obtain the upper and lower dual-flow-space wall slopes at these  $(LDFSF - LDFSS + 1)$  points.

b. General Dual-Flow-Space Wall (NDFS=2). An arbitrary wall contour is specified by tabular input (LDFS - LDFS + 1) y coordinates and the negative of their slopes are specified by the arrays YU and NXNYU for the upper dual-flow-space wall and YL and NXNYL for the lower, respectively. The y coordinates correspond to the (LDFS - LDFS + 1) x coordinates given by XP(LDFS) to XP(LDFS) in Namelist VCL.

#### Namelist VCL

This namelist specifies the variable grid coordinates as well as the parameters that control the sub-cycle and Quick Solver options. For equal or uniform grid spacing, this namelist is left blank. The sub-cycle option allows the part of the mesh with the small grid spacing to be computed for many time steps with the required small time step, whereas the rest of the mesh is calculated only one time step. The Quick Solver option can be used with the sub-cycle option to increase the time step in the small grid part of the mesh and, therefore, reduce the number of time steps or sub-cycles. The Quick Solver allows the increased time step by a procedure that removes the sound speed from the usual C-F-L stability condition. The Quick Solver assumes the flow in the y direction is subsonic. None of the variables specified by this namelist are used by the options necessary to generate nozzle exit plane flow properties for the PEP code so that a blank namelist is input.

Parameter	Description	Default
IST	An integer that, when nonzero, specifies that both the x and y coordinates will have variable grid spacing. When IST=0, the program will generate equally spaced values of XP and YI.	0
XP	A 1D array that denotes the x coordinate grid spacing. The elements of XP begin with the inlet (L=1) and extend to the outlet (L=LMAX). The first element XP(1) must equal XI or XWI(1) of Namelist GEMTRY and XP(LMAX) must equal XE or XWI(NWPTS). For IST not equal to 0, the default values of XP consist of LMAX equally spaced grid points. For IST not equal 0, no default values are given.	None
YI	A 1D array that specifies the y coordinate grid spacing at the inlet or x=XP(1) column of grid points. The elements of YI begin with the centerline or and extend to the wall. If MDFS not equal to 0 and LDFS=1 (Namelist DFLS), then YI(MDFS) must equal YU(1) and a value of YI=YL(1) is not read in. The grid spacing for the columns corresponding to x=XP(2), XP(3),....., XP(LMAX) is proportional to the YI spacing. For IST = 0, the default values of YI consist of MMAX equally spaced grid points. For IST not equal to 0, no default values are given.	YI None
MVCB, MVCT	Integers that, when nonzero, denote which grid points will be sub-cycled. The sub-cycled grid points are M = MVCB to MVCT for all L. The restrictions are: MVCB not equal to 2, MVCT not equal to MMAX-1, and MVCT>MVCB+1. Where dual-flow-space walls are present, MVCB = MDFS + 1 and MVCT=MDFS-1. Finally, if the sub-cycled grid points extend on each side of the dual-flow-space walls, MVCB < MDFS - 1 and MVCT > MDFS + 1.	0 0
NVCM	An integer that, when nonzero, specifies the number of times the small spacing grid points are sub-cycled. If NVCM=0, the program determines the value internally. NVCM must be an odd integer for indexing reasons. See NIQSS and NIQSF for additional details.	0

## Namelist VCL(Cont.)

Parameter	Description	Default
IQS	An integer that, when nonzero, specifies the Quick Solver option. The option assumes that the flow in the y direction is subsonic. Also, if MVCT=MMAX, then the wall boundary must be a no-slip solid wall (TWALL=0 and NOSLIP=1 in Namelist BC). If MVCB=1, then the centerbody boundary must be a no-slip solid wall (NGCB=1 in Namelist GCBL and NOSLIP=1). If dual-flow-space walls are present (see Namelist DFSL), the Quick Solver assumes that the sub-cycled grid points extend on each side of the dual-flow space walls; that is, MVCB < MDFS < MVCT.	0
NIQSS, NIQSF	Integers that, when nonzero, denote at which time step N the Quick Solver will start(NIQSS) and stop (NIQSF). If NIQSS > 1 and NVCMI is nonzero, then the program internally calculates the number of times to sub-cycle the small spacing grid points for $N < \text{NIQSS}$ and uses NVCMI when $N > \text{NIQSS}$ . The default vales are NIQSS = 2 and NIQSF = NMAX in Namelist CNTRL.	2 NMAX
CQS	A parameter that specifies the convergence tolerance for the iteration that locates the characteristic intersection points in the Quick Solver.	.001
ILLQS	An integer that specifies the maximum number of iterations allowed in locating the characteristic intersection points in the Quick Solver.	30
SQS	The coefficient $C_s$ , in Eqs(47) and (49) of Reference 1, that controls the amount of numerical smoothing necessary to stabilize the Quick Solver.	0.5

## 2.4 PEP/VNAP2 Flowfield Model Verification/Demonstration

Two cases were selected to verify/demonstrate the PEP/VNAP2 model. The following subsections discuss the results of the verification and demonstration cases

### 2.4.1 Verification Case 1

Case 1 is an existing VNAP2 sample case for which results are presented in Reference 1. This case is a standard converging/diverging nozzle(see Figure 4 for the geometry) for which nozzle wall pressure and mach number distributions were measured(7). This is a industry standard case that is typically used to validate transonic nozzle flowfield models. The case was set up and run using the steps presented in section 2.3. Figure 5 presents a comparison of the measured data and the VNAP2 results. While this case is basically inviscid it does demonstrate that the VNAP2 module can accurately predict the transonic region of rocket nozzles. The problem name for this case is "back". The input/output files for this case can be found under the `*/prog/ramp/feb96/work` directory of the version of the code that is contained on a 4mm tar tape, titled - PEP/VNAP2, which was delivered to MSFC/ED32 - Mark D'Agostino.

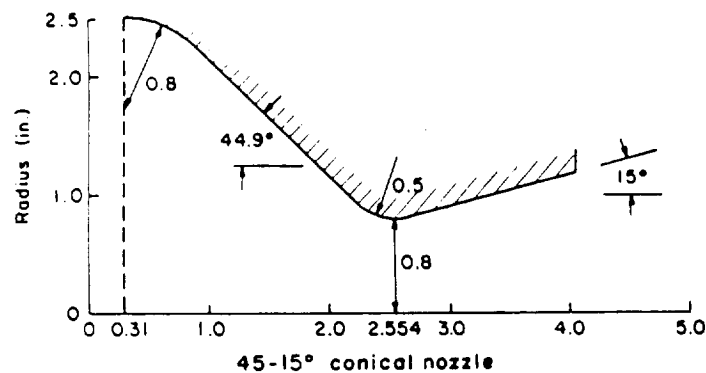


Figure 4 Verification Case Nozzle Geometry

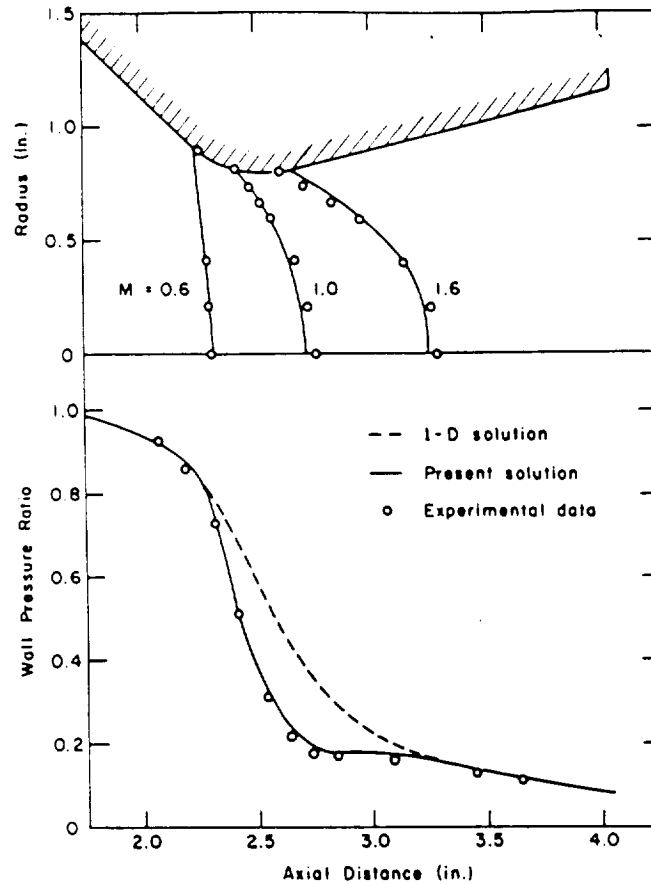


Figure 5 Mach Number Contours and Wall Pressure Ratio for Verification Case

#### 2.4.2 Demonstration Case 1

The Shooting Star solar propulsion demonstration project seeks to demonstrate the practical application of a 1 lbf solar propulsion system. This type of application is exactly what the results of this study is supposed to be able to address. Solar energy is used to heat hydrogen to temperatures of around 5000 degrees R. The heated hydrogen is subsequently exhausted through a high area ratio nozzle ( chamber pressure of approximately 45 psia). This nozzle provides 1 lbf of thrust. The nozzle geometry consists of a 20 degree half angle conical section having a throat diameter of .128 inches and exit diameter of 1.558 inches. The current configuration locates the nozzle in a 20 inch diameter circular duct that has a length of 44 inches. Confining the nozzle and subsequent exhaust in a duct results in potential effects on the design and performance of the system. First, there are plume induced pressure and heat loads on the inside surface of the duct or any other structures that are impinged upon by the exhaust plume. Second

there is a possibility of thrust enhancement or loss due to impingement loads or flow reversal. The Shooting Star solar thruster propulsion system was chosen as a demonstration case for the viscous nozzle/plume option of the PEP code. The new VNAP2-PEP flowfield modeling option was applied to the characterization of the Shooting Star solar propulsion system exhaust plume. The results of solar propulsion system characterization are shown in Figures 6-12. Figures 6-8 show mach number, temperature and pressure contours in the VNAP2 nozzle flowfield. Figures 9-12 show mach number, temperature, pressure and mass flow streamlines in the resulting exhaust plume flowfield calculated using the updated RAMP flowfield module in PEP.

Plume impingement pressure and heating loads were determined for the inside face of the 20 inch diameter duct utilizing the Plume Impingement Code(PLIMP,5). The resulting pressure and heating rate distributions on the cylinder are shown in Figures 13 and 14. These figures show the axial variation of impact pressure and heating rate as a function of the location of the exit plane of the nozzle in the duct. If the nozzle were to be located at the furthest downstream extent of the duct then only that portion of the distribution upstream of the nozzle exit would be applied to the duct. Conversely, if the nozzle were located at the entrance of the duct only the downstream loads would be applied to the duct. If the nozzle were located at some point in the nozzle then impingement loads would be produced both upstream and downstream of the nozzle exit. In the event the nozzle was located at some point inside the duct then there is a potential of thrust cancellation due to the reversal of exhaust upstream of the exit. This is due to the fact that the impingement flowfield on the inside of the duct produces an adverse pressure gradient along the nozzle(see Figure 13). The peak pressure occurs near 1.2 feet downstream of the exit. Thus all mass upstream of this location would be reversed. Examination of the mass flow streamlines of Figure 12 shows that a maximum of 20 % of the total mass can be reversed. The total mass flowing through the nozzle is .001185 lbm/sec.

One of the potential fixes for retaining lost thrust is to place a circular plate at the exit of the nozzle so that no mass can be reversed upstream. The presence of the plate will result in an net increase of .0001 lbf of thrust. The resultant radial distribution of pressure and heating loads on the circular plate are presented in Figures 19 and 20. The plate is assumed to be located .25 inches upstream of the lip. Pressure and heating rate distributions are presented as a function of radial distance away from the nozzle centerline.

The problem name for this case is "sstar". The input/output files for this case can be found under the \*/prog/ramp/feb96/work directory of the version of the code that is contained on a 4mm tar tape, titled - PEP/VNAP2, which was delivered to MSFC/ED32 - Mark D'Agostino.

**FIGURE 6 MACH NUMBER CONTOURS  
FOR 1 LBF SOLAR THRUSTER NOZZLE(PC=45 PSIA, TC=5000 R)**

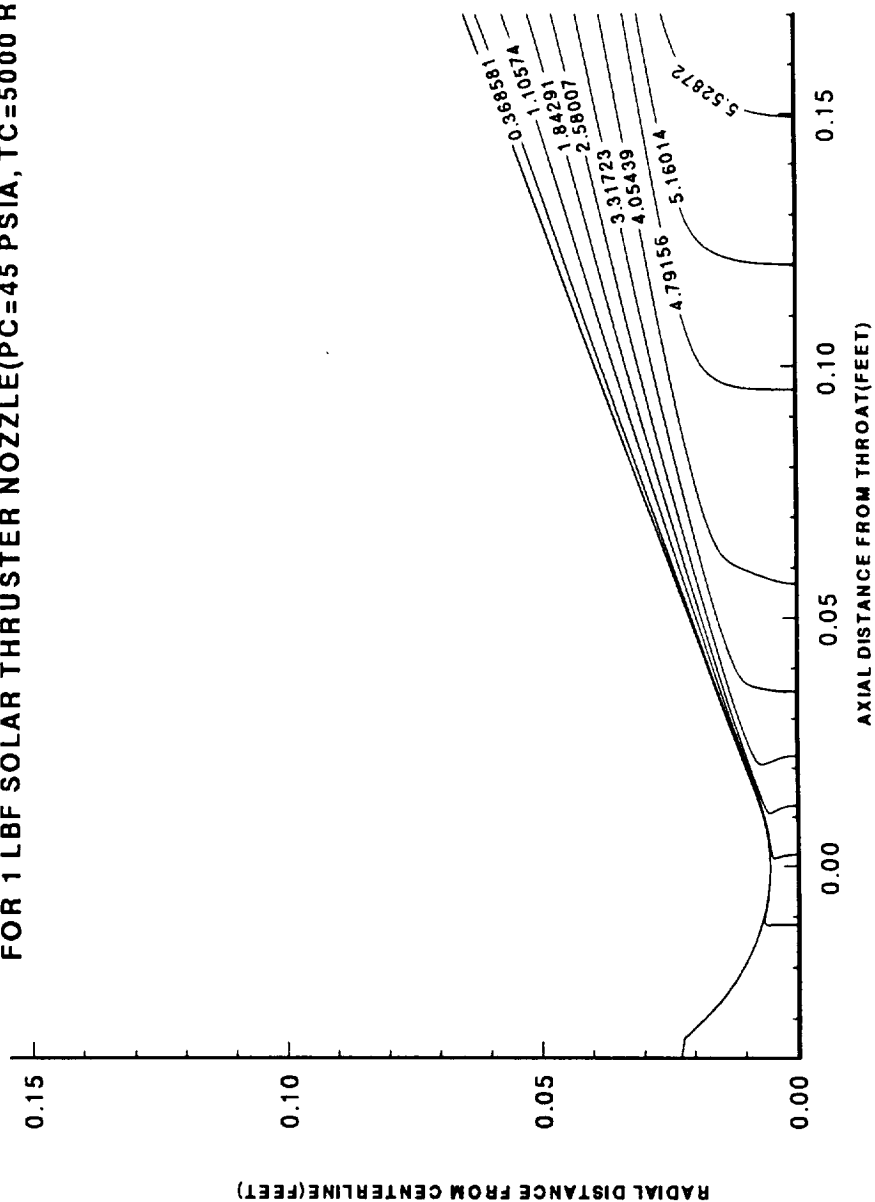
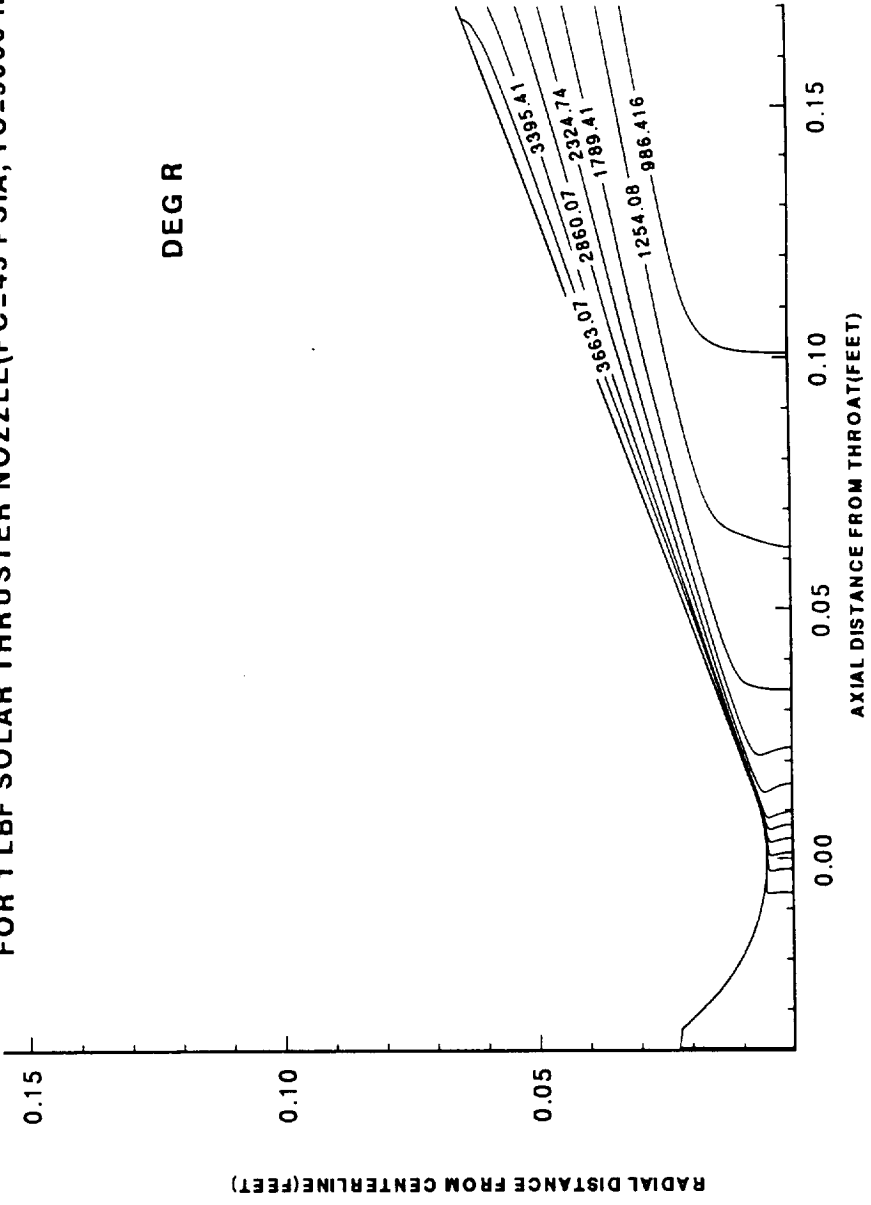
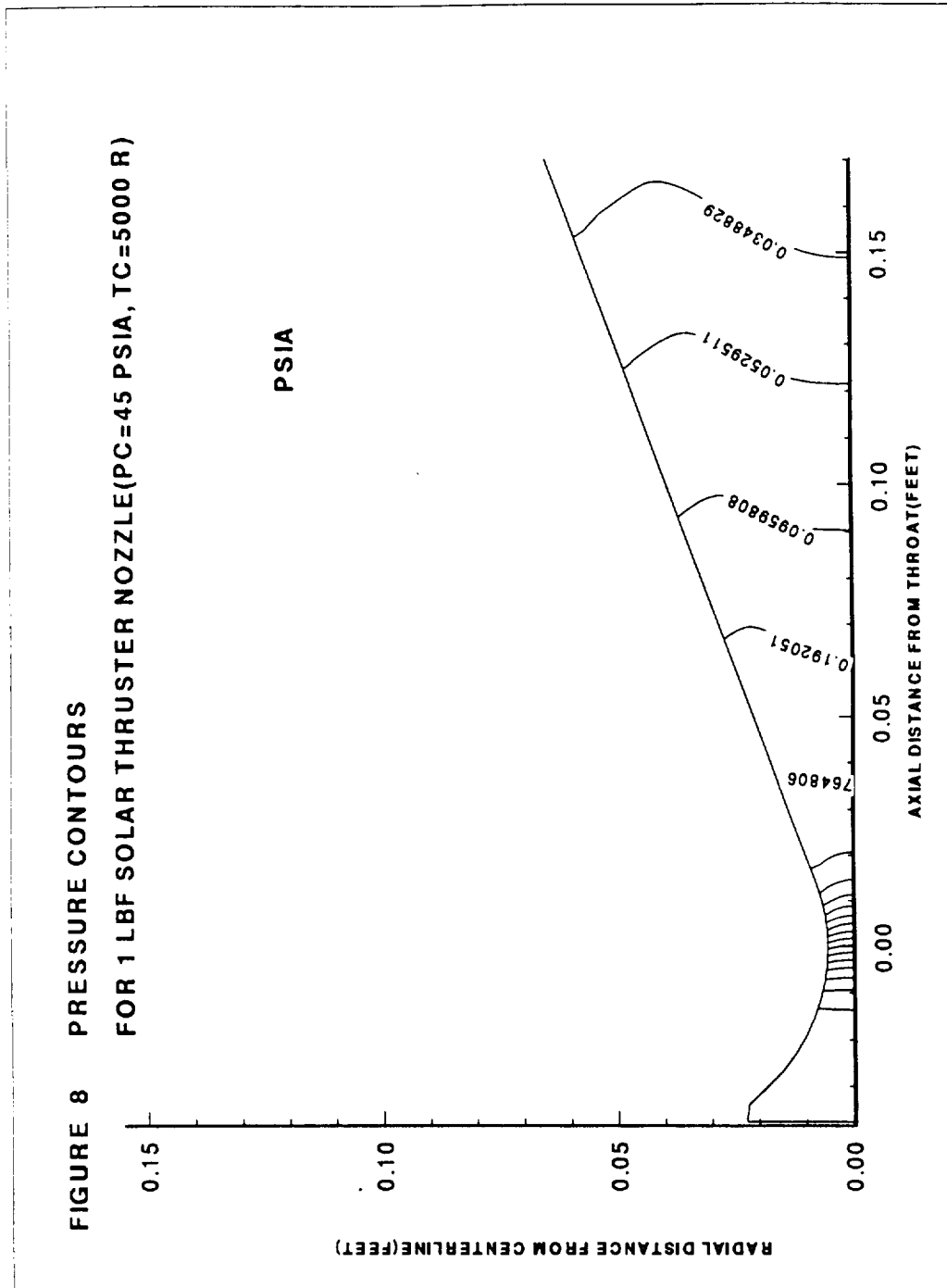
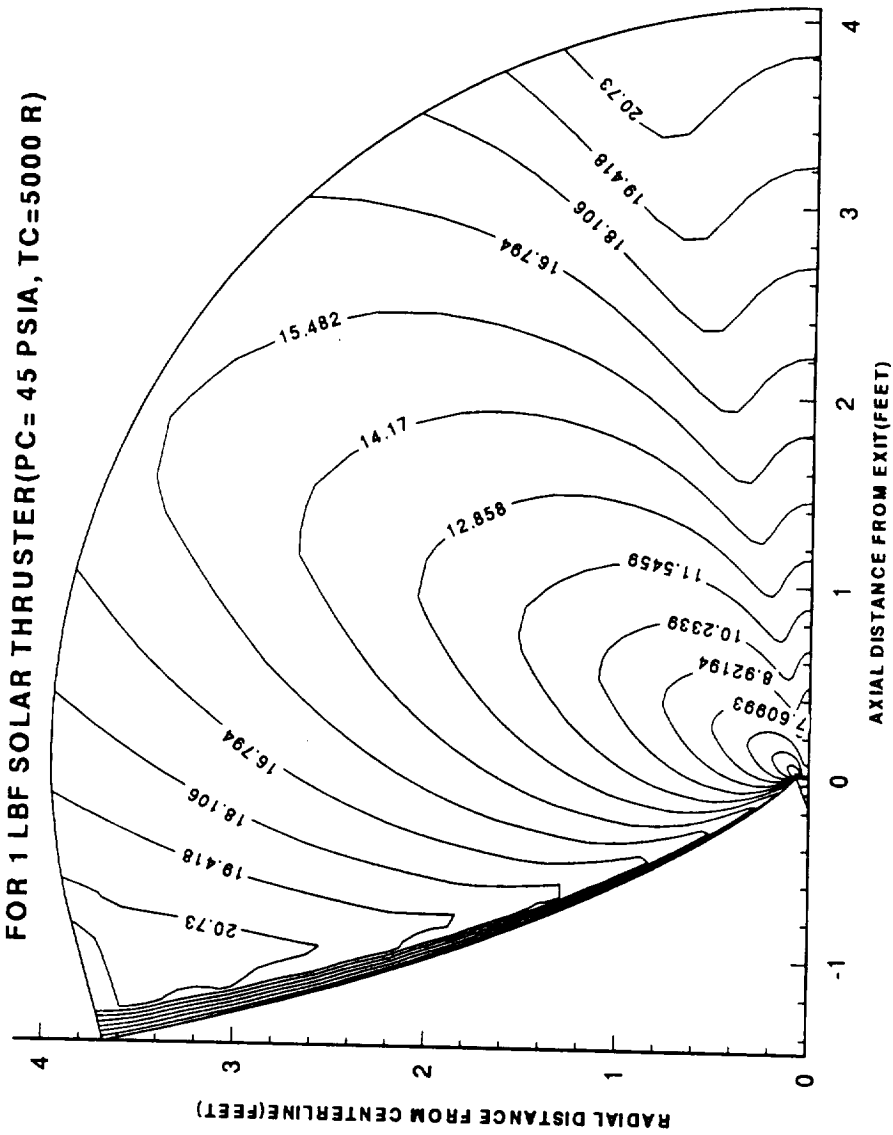


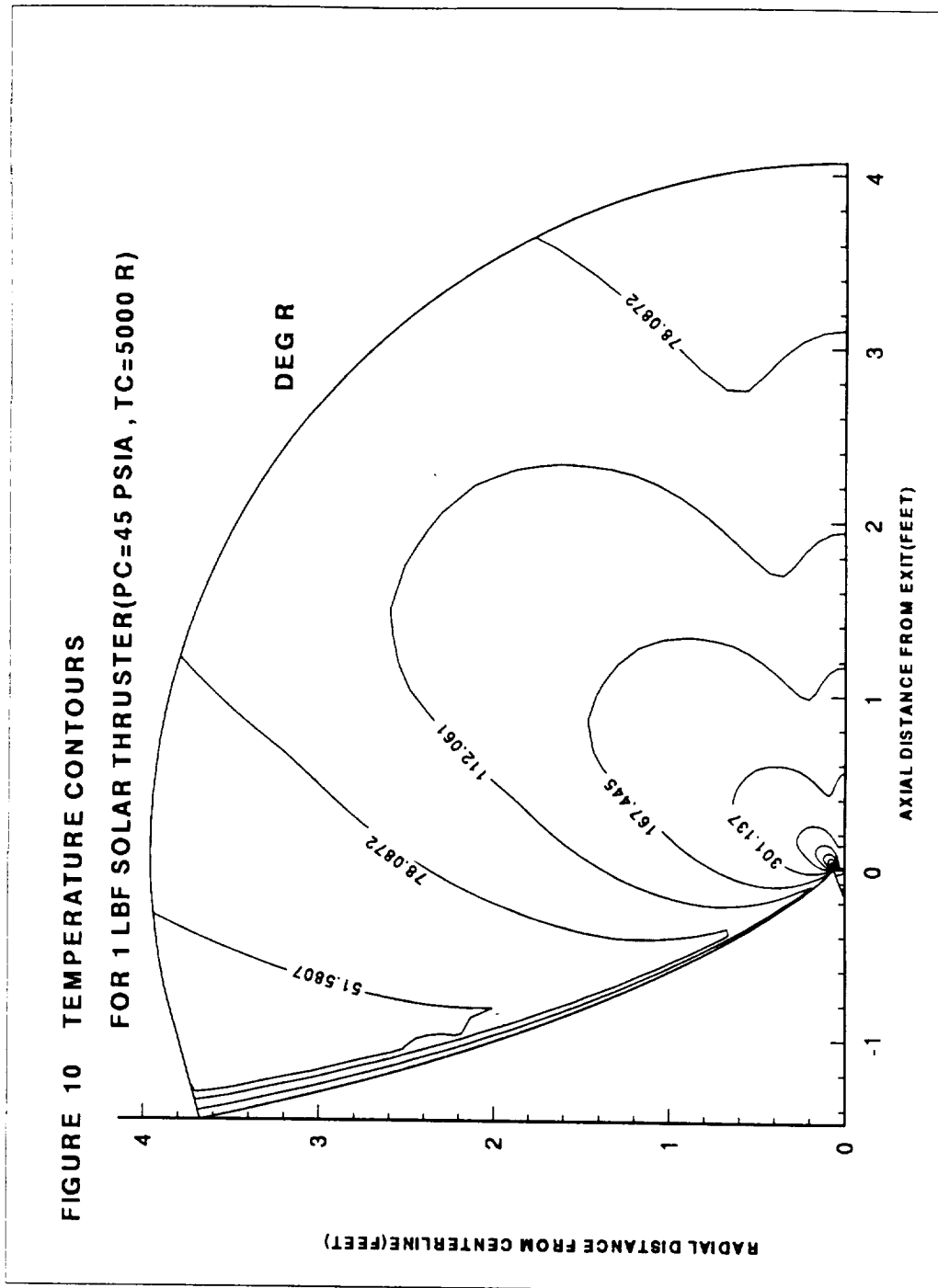
FIGURE 7 TEMPERATURE CONTOURS  
FOR 1 LBF SOLAR THRUSTER NOZZLE(PC=45 PSIA, TC=5000 R)



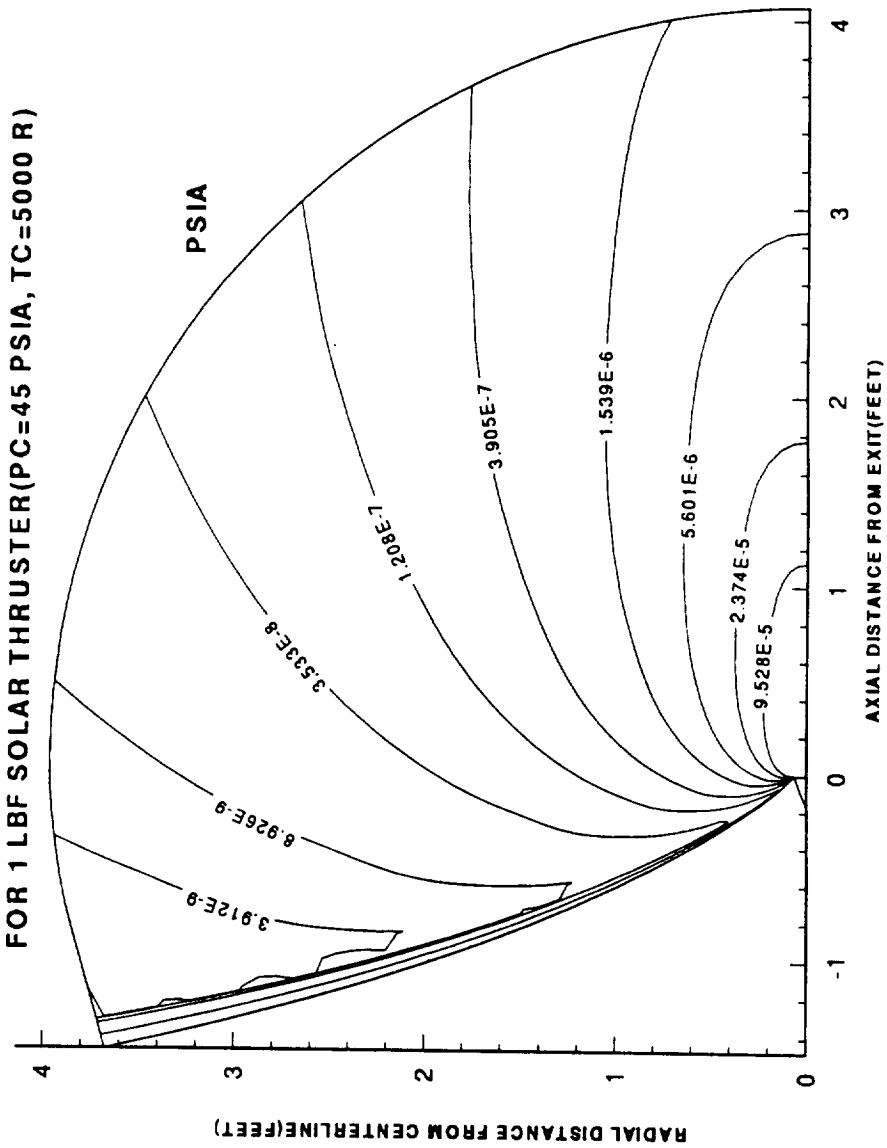


**FIGURE 9 MACH NUMBER CONTOURS  
FOR 1 LBF SOLAR THRUSTER(PC= 45 PSIA, TC=5000 R)**

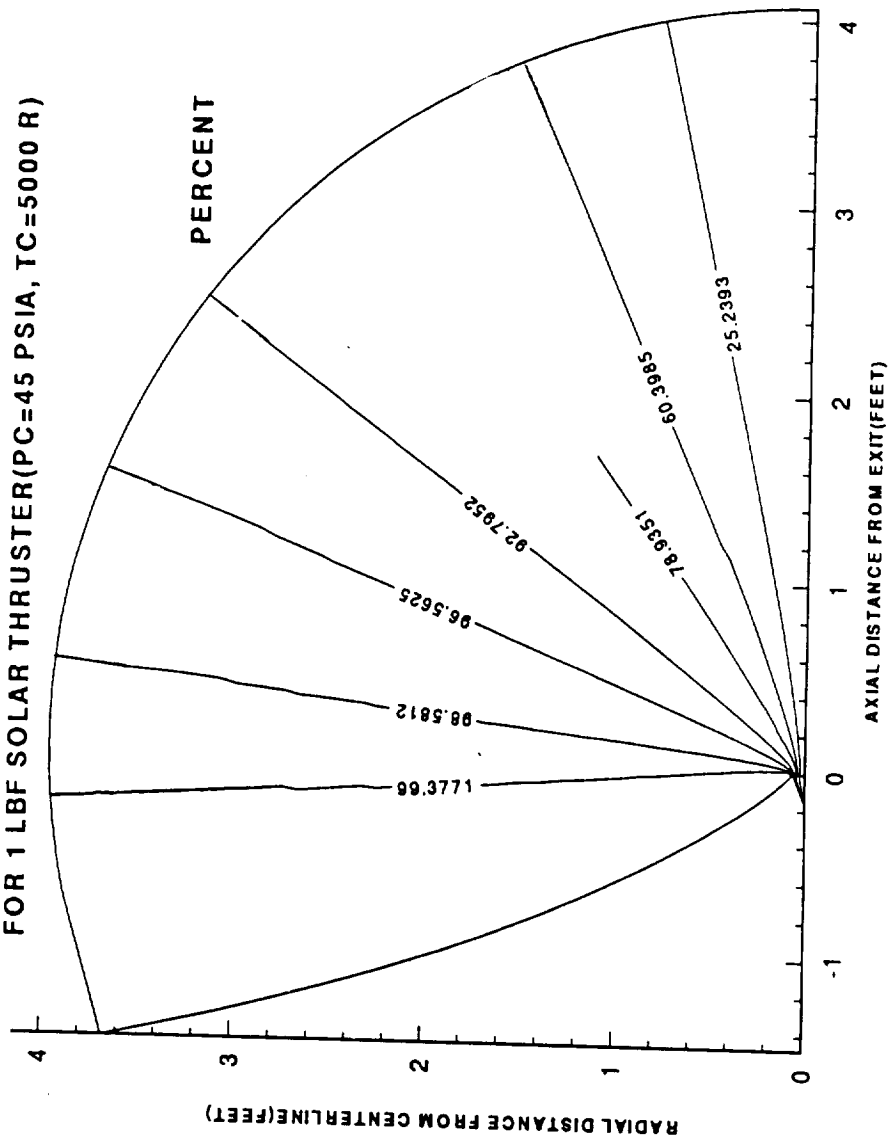




**FIGURE 11 PRESSURE CONTOURS  
FOR 1 LBF SOLAR THRUSTER(PC=45 PSIA, TC=5000 R)**



**FIGURE 12 MASS FLOW STREAMLINES  
FOR 1 LBF SOLAR THRUSTER(PC=45 PSIA, TC=5000 R)**



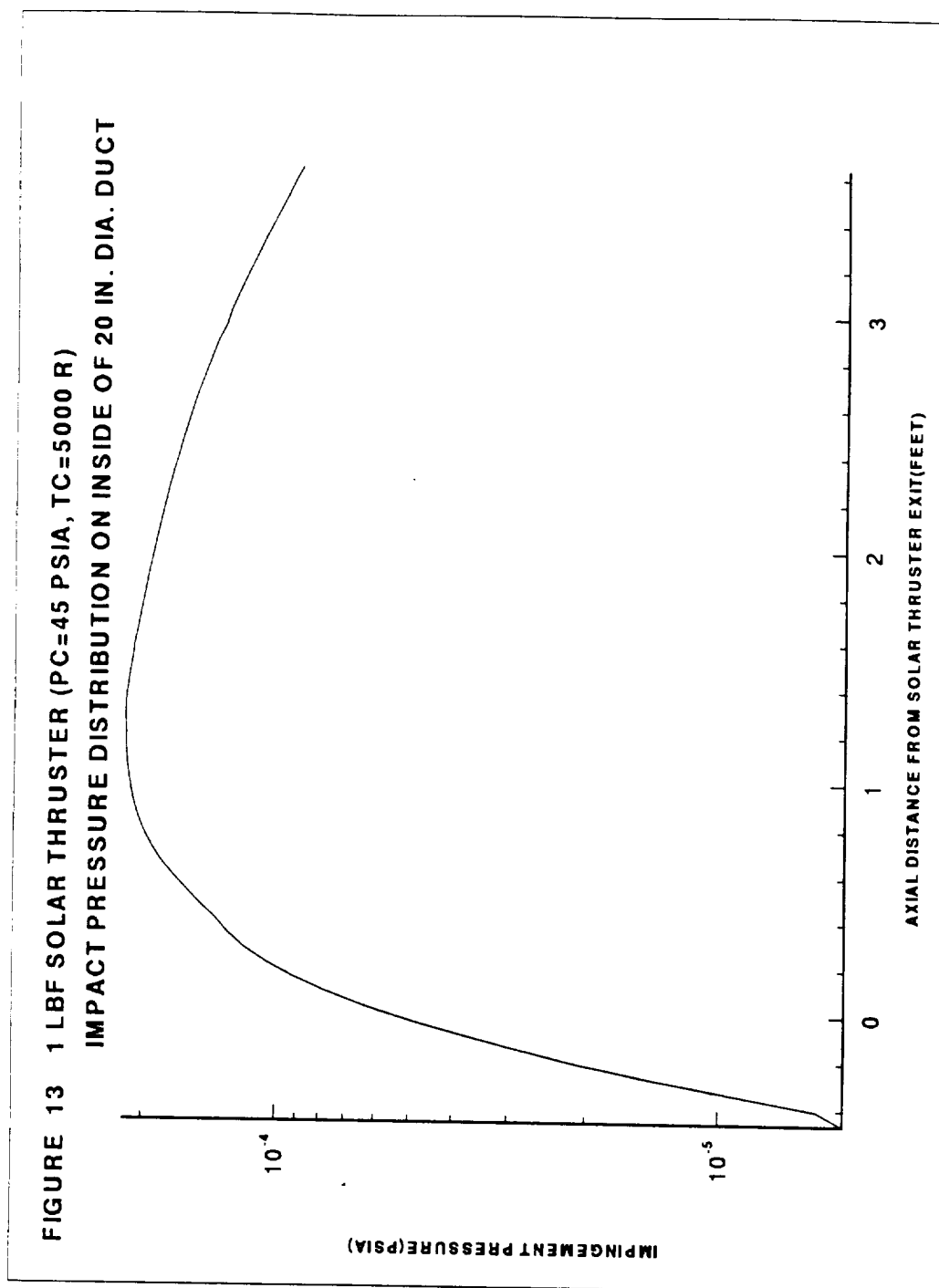
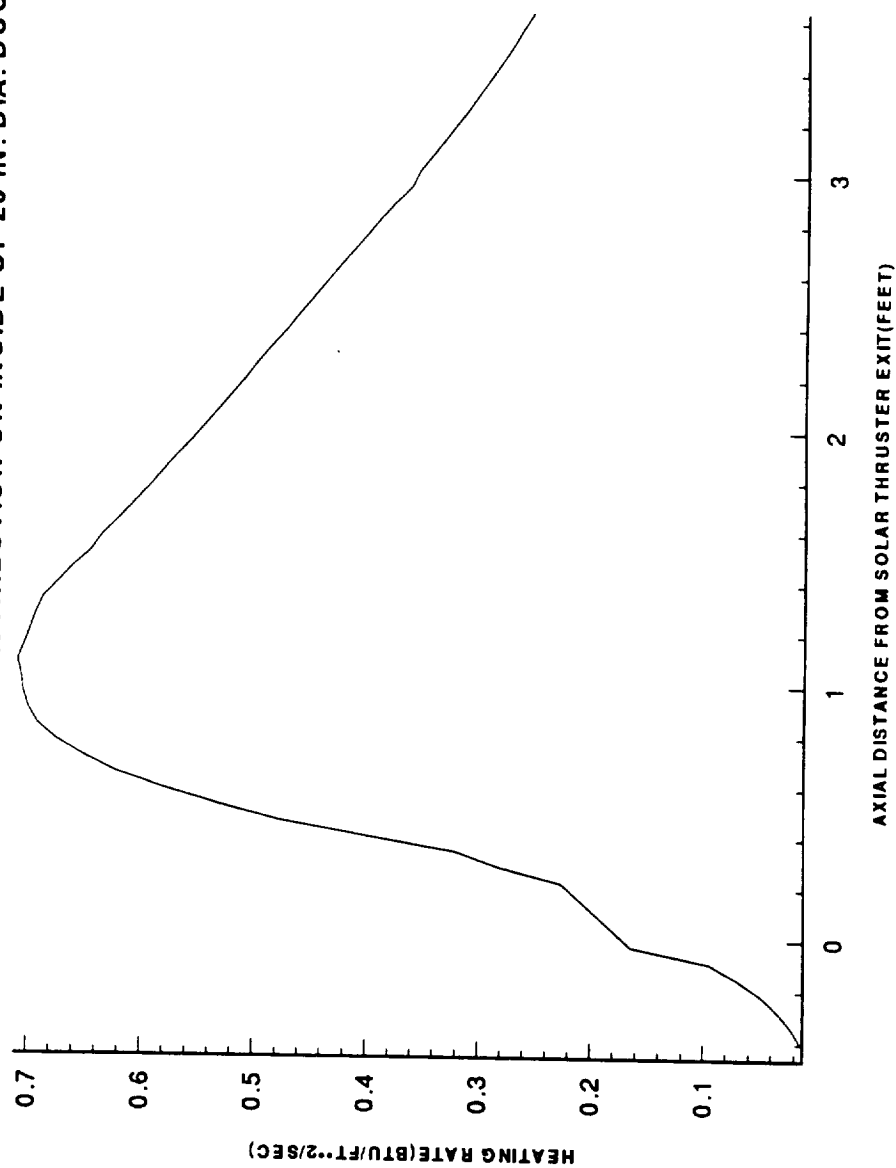


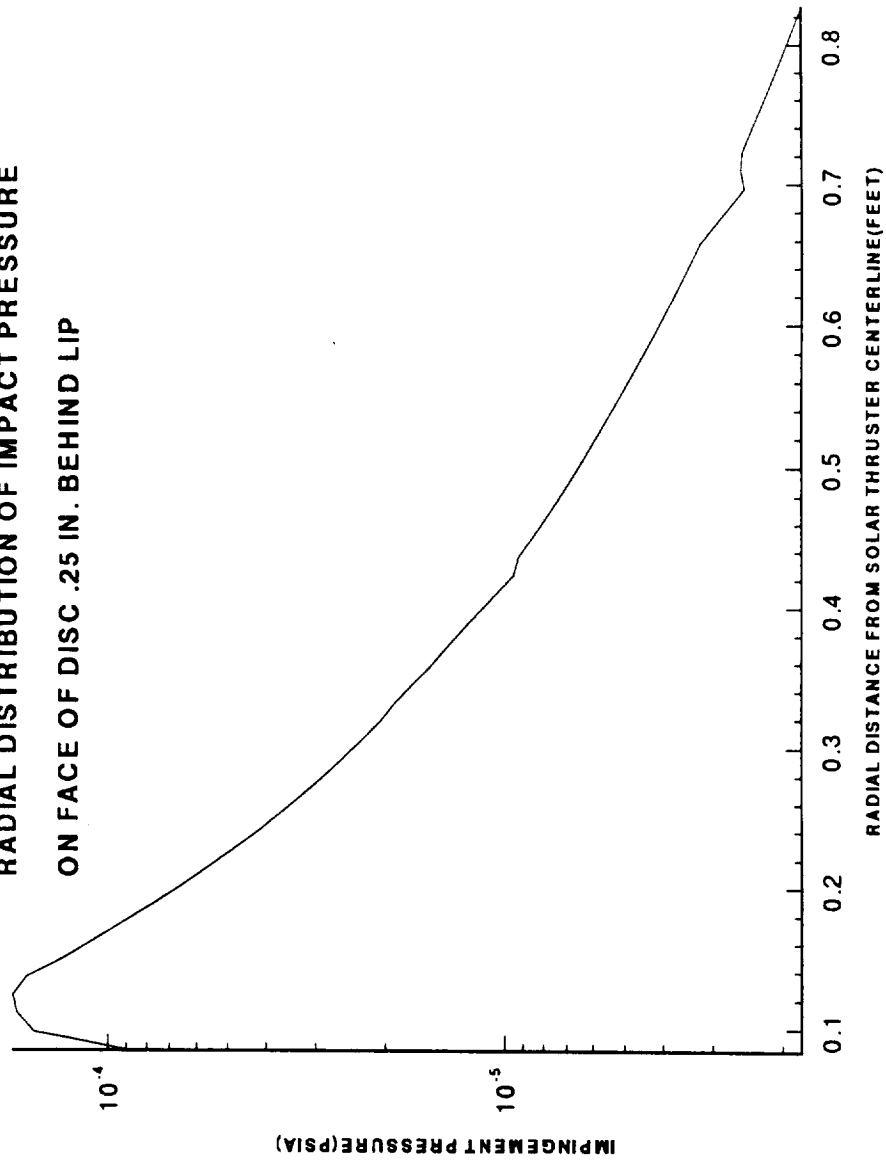
FIGURE 14 1 LBF SOLAR THRUSTER (PC=45 PSIA, TC=5000 R)  
HEATING RATE DISTRIBUTION ON INSIDE OF 20 IN. DIA. DUCT



**FIGURE 15 1 LBF SOLAR THRUSTER(PC=45 PSIA, TC=5000 R)**

**RADIAL DISTRIBUTION OF IMPACT PRESSURE**

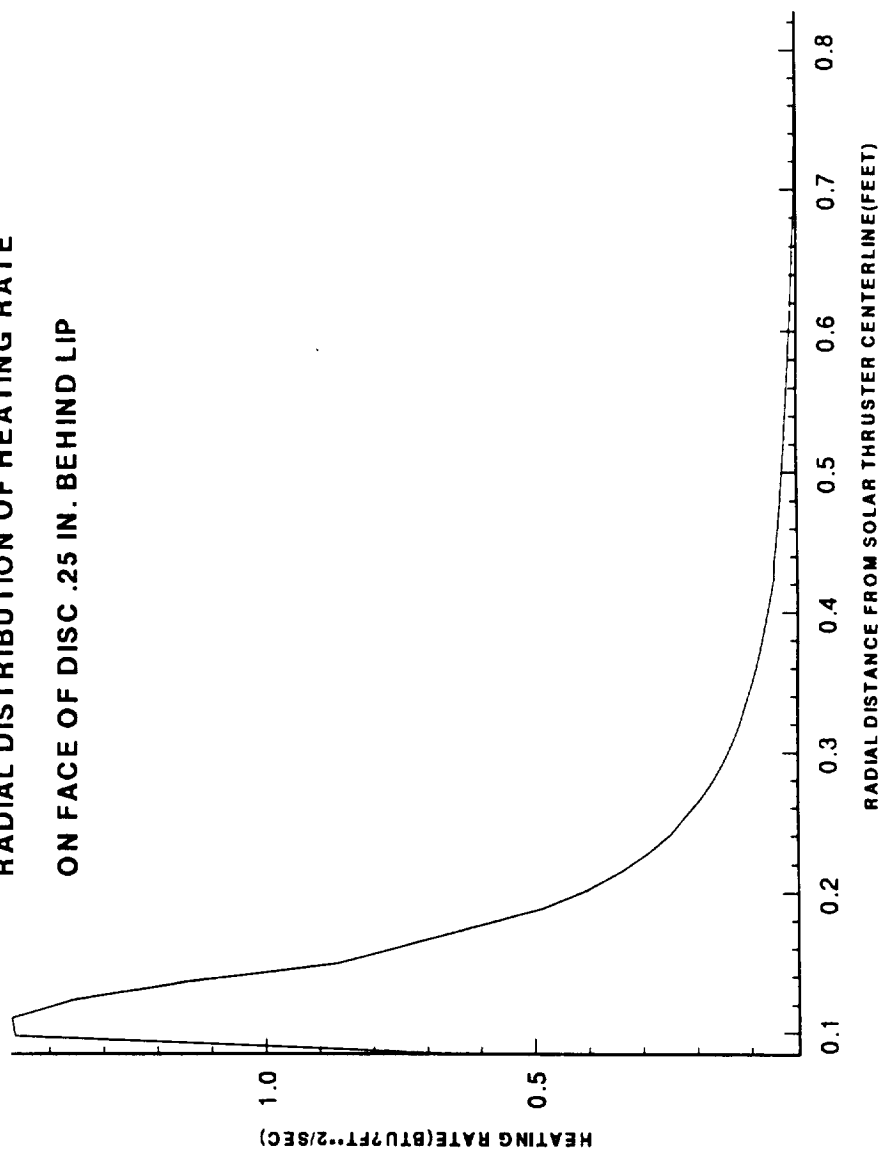
**ON FACE OF DISC .25 IN. BEHIND LIP**



**FIGURE 16 1 LBF SOLAR THRUSTER(PC=45 PSIA, TC=5000 R)**

**RADIAL DISTRIBUTION OF HEATING RATE**

**ON FACE OF DISC .25 IN. BEHIND LIP**



### 3.0 CONCLUSIONS AND RECOMMENDATIONS

This report presents the results of a study to develop an accurate, practical and user friendly method of calculating high altitude rocket exhaust plume flowfields emanating from highly viscous, low chamber pressure, high area ratio rocket motor nozzles. This report summarizes the results of this study, provides instruction in the use of the VNAP2/PEP code, describes the input data variables and presents validation and demonstration case results. The author would like feedback from users on any problems that are encountered or suggestions on improvements of this version of the PEP model. The deliverable version of the code is contained on a 4mm tar tape, titled - PEP/VNAP2. This tape was delivered to MSFC/ED32 - Mark D'Agostino and installed on his SGI system..

### 4.0 REFERENCES

1. Cline, Michael C., "VNAP2: A Computer Program for Computation of Two-Dimensional, Rime-Dependent, Compressible, Turbulent Flow", LA8872/UC-32, Los Alamos National Laboratory, Los Alamos, New Mexico, August 1981.
2. Kawasaki, A.H., et. Al., "Viscous Interaction Performance Evaluation Routine for Two-Phase Nozzle Flows with Finite Rate Chemistry", Phillips Laboratory Report, PL-TR-92-3053, January 1993.
3. Smith, S.D., "Update to the RAMP2 Computer Program," SECA-FR-93-19, SECA, Inc., Huntsville, AL., December 1994.
4. Smith, S.D., "Artificial Intelligence in Rocket Exhaust Plume and Plume Environments for Launch Vehicles and Spacecraft Design", HSC-TR95-01, Huntsville Sciences Corporation, Huntsville, AL., June 1995.
5. Smith, S.D., "Model Development for Exhaust Plume Effects on Launch Stand Design-PLIMP/LSD," SECA 93-FR-9, SECA, Inc., Huntsville, AL, June 1993.
6. R.F. Cuffel, L.H. Back, and P.F. Massier, "Transonic Flow-Field in a Supersonic Nozzle with Small Throat Radius of Curvature," AIAA J. 7.1364(1969).

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
1. AGENCY USE ONLY (Leave Blank)		2. REPORT DATE 2/07/98	3. REPORT TYPE AND DATES COVERED	
4. TITLE AND SUBTITLE "Advanced Space Propulsion System Flowfield Modeling"			5. FUNDING NUMBERS	
6. AUTHOR(S) Sheldon D. Smith				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Huntsville Sciences Corporation 7525 S. Memorial Pkwy, Suite D Huntsville, AL 35802			8. PERFORMING ORGANIZATION REPORT NUMBER HSC-FR-98-2	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) NASA/MSFC			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NAS8 - 40845	
11. SUPPLEMENTARY NOTES				
12a. DISTRIBUTION / AVAILABILITY STATEMENT Unlimited			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) Solar thermal upper stage propulsion systems currently under development utilize small low chamber pressure/high area ratio nozzles. Consequently, the resulting flow in the nozzle is highly viscous, with the boundary layer flow comprising a significant fraction of the total nozzle flow area. Conventional uncoupled flow methods which treat the nozzle boundary layer and inviscid flowfield separately by combining the two calculations via the influence of the boundary layer displacement thickness on the inviscid flowfield are not accurate enough to adequately treat highly viscous nozzles. Navier Stokes models such as VNAP2 can treat these flowfields but cannot perform a vacuum plume expansion for applications where the exhaust plume produces induced environments on adjacent structures. This study built upon recently developed artificial intelligence methods and user interface methodologies to couple the VNAP2 model for treating viscous nozzle flowfields with a vacuum plume flowfield model (RAMP2) that is currently a part of the Plume Environment Prediction Model. This study integrated the VNAP2 code into the PEP model to produce an accurate, practical and user friendly tool for calculating highly viscous nozzle and exhaust plume flowfields				
14. SUBJECT TERMS			15. NUMBER OF PAGES 43	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT UNCL	18. SECURITY CLASSIFICATION OF THIS PAGE UNCL	19. SECURITY CLASSIFICATION OF ABSTRACT UNCL	20. LIMITATION OF ABSTRACT None	